Design and CFD analysis of vortex Tube with and without insulation

A Thesis Submitted

In partial fulfilment for the reward of the degree of

Master of technology

In Mechanical Engineering

SUBMITTED BY

Siddharth Ranjan

(2K19/THE/20)

UNDER THE GUIDANCE OF

Prof. B.B. ARORA

DEPARTMENT OF MECHANICAL, PRODUCTION & INDUSTRIAL

AND AUTOMOBILE ENGINEERING

DELHI TECHNOLOGICAL UNIVERSITY BAWANA ROAD, DELHI-110042

CANDIDATE'S DECLARATION

I, SIDDHARTH RANJAN, here by certify that the work which is being presented in thesis entitled "Design and CFD analysis of vortex Tube with and without insulation" being submitted by me is an authentic record of my own work carried out under the supervision of Prof B.B. Arora Department of Mechanical Engineering, Delhi Technological University Delhi.

The matter presented in this thesis has not been submitted in any other University/Institute for the award of M.Tech Degree.

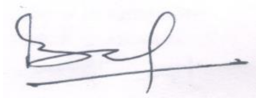
SIDDHARTH RANJAN

(2K19/THE/20)

CERTIFICATE

I, SIDDHARTH RANJAN, hereby rectify that the work which is being presented in this thesis entitled “Design and CFD analysis of vortex Tube with and without insulation” in the partial fulfilment of requirement for the reward of degree of Masters of Technology in Thermal Engineering submitted in the Department of Mechanical Engineering, Delhi Technological University Delhi is an authentic record of my own work carried out during a period from July2020 to June 2021, under the supervision of Prof. B.B. Arora, Department of Mechanical Engineering, Delhi Technological University Delhi.

The matter presented in this thesis has not been submitted in any other University/Institute for the award of M.Tech Degree.



Prof B.B. ARORA

SUPERVISOR

Department of Mechanical Engineering

Delhi Technological University, Delhi

ACKNOWLEDGEMENT

First and foremost, praise and thanks goes to my God for the blessing that has bestowed upon me in all my endeavours.

I am profoundly thankful of the inspiration, direction, titillation and patience of Prof B.B.ARORA, professor, my advisor and guide. I appreciate his wide range of experience and attention to detail, as well as his consistent support over the years. It should not be mentioned that a large part of this study is a consequence of collaborative collaboration, without which it would not have been possible to complete the work.

I sincerely thank Prof B.B. ARORA for his guidance and relentless support over the year. I am thankful for his worthy suggestions and timely cooperation during his work on the project.

I would like to extend my gratitude to Prof. S. K. Garg, Head, Mechanical Engineering Department for providing this opportunity to carry out this present work.

I would like to thank my family members in this occasion for their unqualified affection, moral support and motivation to complete this project in due course.

Siddharth Ranjan

(2K19/THE/20)

ABSTRACT

Industries replace failing tools with new ones without analysing and resolving the root cause of failure. To avoid cutting tool failure owing to thermal shocks and roost development, it is necessary to enhance the tool's quality. On the other hand, the vortex cooling system guarantees that the instrument is devoid of the aforementioned faults. The temperature distribution in the vortex tube's axial direction was investigated, and it was determined to be sub-zero temperatures suitable for cooling. To model flow patterns, thermal separation, and pressure gradient phenomena, CFD methods are employed. CFD is utilised in this research to determine the optimal design of the vortex tube.

Experimental studies are performed to investigate the effect of geometrical parameterson temperature separation and flow field in the vortex tube. Inlet dimensions, length and diameter of vortex tube are the main geometrical parameterstakenintoaccount. Two types of Vortex tube model designed here one with insulation of wooden material and other one is without insulation; Experiments show that most of the temperature separation happens near inlet from the studies on length of vortex tubes. Inlet nozzle with lower aspect ratio gives better temperature separation. Temperature separation is further increased when convergent divergent nozzle is used. Flow field studies on different geometrical parameters show that number of helical turns and residence time increase with temperature separation. Experimental studies on scale effect in mass flow rate between small and big vortex tubes are discussed. It is found that big vortex tube requires seven times the mass flow rate needed by the small vortex tube.To analyse fluid behaviour within the vortex tube, a numerical analysis was conducted utilising different geometrical and thermal physical factors. ANSYS FLUENT 15.0 was used to solve the governing equations using a 3D model in a fluid domain. The design proved to be a model for others to follow.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

CHAPTER 1

INTRODUCTION

1.1. Introduction

The vortex tube is a simple mechanical device that splits a compressed gas stream into cold and warm streams without the need of chemical processes or an external source of energy. This device divides a compressed isothermal gas flow into two separate temperature flows. When compared to traditional cooling and heating equipment, the advantages include being simple, small, and lightweight, being low-cost, being electrical or chemical-free, and having longer service periods. A central orifice adjacent to the input pin allows cold gas to flow through the tube, while hot gas flows towards the valve regulator and leaves the tube.

The vortex tube was invented in 1933 by French physicist Georges J. Ranque, who published a lengthy paper on it in 1947, and called it the Web Tube in honour of his mother. Rudolf Hilsch, a German physicist, developed the idea and presented it in his book (literally, whirl pipe). Linderstrom-Lang developed the vortex tube in 1967 to separate gases, oxygen and nitrogen mixtures, carbon dioxide and helium, carbon dioxide and air, and carbon dioxide and nitrogen. Interestingly, in a laboratory experiment in which free body rotation is generated from the core and there is a thick boundary layer on the wall, Vortex tubes seem to work with fluids to a certain extent. The separation of air enables the exhaust to flow cooler, similar to that of a refrigerator. It is possible to see the thermal energy separation process in incompressible (liquid) vortex flow when the intake pressure is high, for example, between 20 and 50 bar.

Injected into the vortex chamber by the intake pins, a whirling current is formed when high pressure gas (6 bar) is tangentially injected. It extends and cools if the gas swirls to the centre of the chamber. Some of the gas passes through the whirlpool chamber, while others travel by the cold exhaust immediately. A portion of the gas flows through the vortex tube, reversing the speed axis from the hot to the cold end of the tube. The gas leaves in warm exhaustion at higher temperatures, while the gas leaves in cold exhaust at lower temperatures. [11]

A Vortex tube offers the following advantages over traditional commercial cooling devices: single, no moving parts, no energy or chemicals, small and lighter, free of charge, immediate cold air, long lasting (thanks to stainless steel and clean working medium), adjustable temperature. Its low thermal efficiency, on the other hand, is a significant stumbling block. Loudness and availability may also limit the usage of compressed gas. RHVT becomes a great instrument for gas heating, cooling gas, gas purification, drying gas mixtures, gas separation, DNA applications, natural gas liquefying, and more when the main drivers are compactness, reliability, and lower equipment costs, and operating efficiency is decreased. [1]

1.2 Principle and Theory

The vortex tube, sometimes known as a Ranque-Hilsch vortex tube, has a theoretical history that goes back to the 1930s, when a prototype was originally constructed by French physicist George Ranque. When the German army occupied most of France in 1945, Rudolf Hilsch, a German scientist, improved his design to create a better version of the tube.

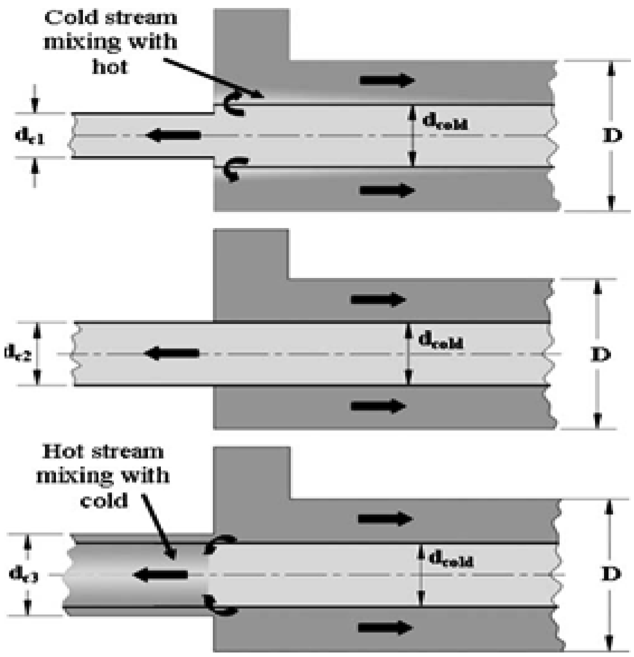


Figure 1.1: Flow Pattern Inside the Vortex Tube

The tube has been given many names by inventors, although it is most often attributed to Hilsch, who has made major contributions to the vortex tube's improvement. The compressed air inlet is located in the middle of the tube. The inlet is much closer to the cold outlet than the hot outlet. With this function, a very important aspect of the tube will be addressed very soon. The spiral chamber is mentioned in the middle of this section. This spiral chamber is an essential component of the tube since it is responsible for the gas heating and cooling separation. As illustrated in Figure 1.1, it is primarily based on rotational motion mechanics and Maxwell's random distribution law.

When compressed operating fluid is injected tangentially into the vortex tube, it creates an angular momentum in the working fluid's linear momentum. Because of its central flight characteristics, peripheral fluid entering the ring area is more angular, i.e. nematogenic, than fluid in the centre zone. As a result, the tube wall has a higher temperature than the centre.

Maxwell's Law States serve as the basis for the kinetic theory of gases, which aids in the understanding of fundamental gas properties such as diffusion and pressure. The rule is most often applied to speed, but it may also be applied to molecular dynamics. We're focusing on the speeds of all spiral chamber molecules right now. The so-called Maxwell-Boltzmann distribution functions, which are essential in physics and chemistry.

Because their speed is greater than the initial speed at which the gas enters the tube, these molecules have more kinematic energy than the other molecules. These are more technologically advanced compounds. There are no power sources present, nor is any activity taking place on compressed air, which leaves the only possible explanation being that internal molecular energy being transformed to film energy. To show that the tube did not violate the laws of thermodynamics, the air had to be compressed somewhere upstream from where the tube was being tested.

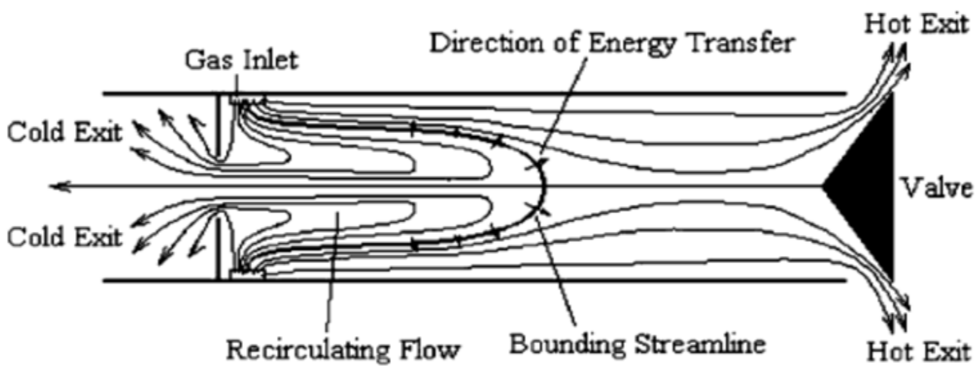


Figure 1.2: Energy Separation in a Counter Flow Vortex Tube

The first thermodynamic law says that a thermodynamic system's gain in internal power equalled the quantity of heat energy supplied to the system minus its work on the system via its surroundings.

Mathematically, this law is

$$U = Q - W \qquad \text{Eqn (1)}$$

Or, written in differential form,

$$dU = dQ - dW \qquad \text{Eqn (2)}$$

In the equations, U refers to the system's internal energy, Q to the system's heat, and W to the system's working environment. The thermodynamic law lowers to a level by applying equation[1] to the vortex tube [3] below:

$$dU = dQ \qquad \text{Eqn (3)}$$

In other words, every change in internal energy is connected to a heat change in the system.

The first rule applies when a gas enters the pipe because, since no system work is done, internal energy must be converted to kinetic energy when it begins spinning and producing the vortex. Figure 1.2 shows a diagram of how the air in the tube moves.

Kinetic energy changes in the molecules after the initial reduction in internal energy occurs and the gas begins to spread over the generated vortex pattern. Because the radius of rotation is greatest in this direction than the molecules at the centre with a radius of zero to spin, their kinetic energy must be considerable when the molecules spread out to the tube wall. The kinetic energy is pushed to the molecules on the tube wall by the diffusion radial pressure gradient. Because of the pressure gradient, the total energy (kinetic and potential) of the pipe walls will be higher than the tube axis molecules, as illustrated in Fig1.2.

If you take a step back, you'll see that the cold-exit valve is considerably closer to the heat outlet. Both the Z and TRI directions begin slowly as the molecules pass through the tube. As the tube falls, the vortex's axial convection diminishes, causing the tube to slow down. When the gas fills up more space and releases pressure, the molecules' diffusion rate slows, causing the molecules to spread more slowly down the line. This is accomplished by utilising less fine energy, which increases the gas's temperature and transforms it to internal energy. Because the original radial pressure gradient resulted in the transfer of kinetic energy to external molecules, the fine energy to be converted back into internal energy is greater, resulting in a higher temperature. The dissipation of kinetic energy refers to the process of converting air flow energy into thermal energy.

Due to a goal to minimise the impacts of viscous kinetic energy dissipation in the cold exit near the intake, this has been implemented. As a result, the exit must be as near as possible to the intake, since the longer a gas is removed from the intake, the more viscous dissipation takes place and the higher the temperature of the gas. Due to the fact that the cold exit is not exactly alongside the intake, it is possible to utilise the radial pressure differential and transfer kinetic energy from the molecules in the tube wall. The outlet is strategically positioned to allow gas to be expelled from the tube's centre. A same rationale as for positioning the cold outlet farther away from the intake may be used to positioning the hot outlet. In order to keep the tube cold, the scientist allowed air to go more slowly down the tube rather than more rapidly down the tube. He warms up as a result of the dissipation of thermal energy in a viscous way. Since of axial convection decreasing as the gas goes down the pipe, it results in a more uniform stream because the temperature gradient and pressure down the pipe are reduced. A result of the rapid gains in kinetic energy, almost all of the stream reaches the outlet at the same temperature and at a higher temperature than it did at the intake point. However, in order to get the highest temperature possible, the exit is moved away from the tube walls.

This experiment is driven by two key assumptions in addition to the concept behind how the vortex tube works. I'm hoping that the first and second laws of thermodynamics hold the key to understanding why the tube does not miraculously create nothing. The first and second laws provide enthalpy and entropy balances.

These are two equations that illustrate how the thermodynamic variables of enthalpy (H) and entropy (S) in the system are kept and maintained. Enthalpy is the amount of thermal energy in whatever form that a substance or system has. Internal energy, system work, and so forth are all part of this. Entropy is a measure of a system's unpredictability, or energy that is not available for use. Both balances, hopefully, will demonstrate that the system does not change the temperature for no reason.

Temperature variations of 30 K (or 10% ambient) may be readily produced with a pressure drop ratio of 5 and ambient temperature, enough for basic cooling.

Temperature fluctuations of 30 K (or 10 percent ambient) with a pressure drop ratio of 5 and environmental temperature may easily be generated, adequate to cool the base.

1.3 Working

How is it possible for a single compressed-air stream to generate both cold and hot air? Many have attempted to explain it, including the French physicist who invented the Vortex Tube in the 1930s, Georges Ranque. Many other theories have been proposed.

Vortex Tubes behave in a highly predictable and repeatable way. You get hot air from one end of the tube and cold air from the other when the vortex generator discharges compressed air into a tube. Figure 1.3 shows how a small valve at the hot end may alter the air output volume and temperature from the cold end, which can be adjusted with the simple control button.

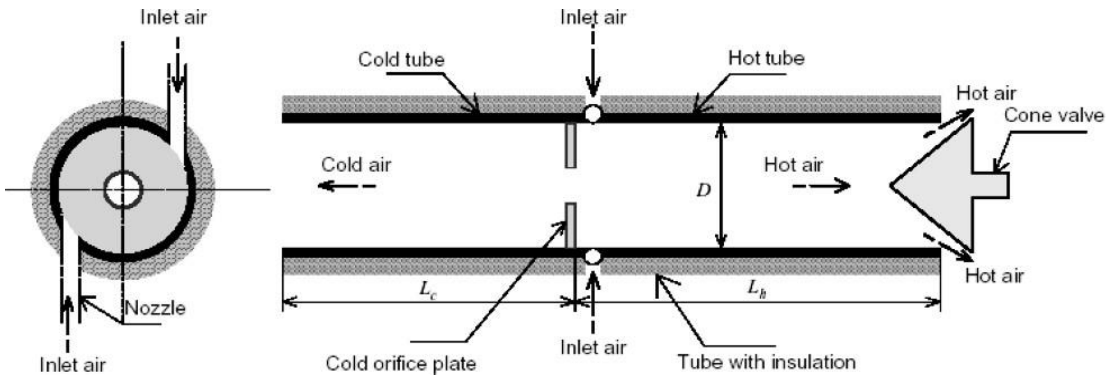


Figure 1.3: Interior of Vortex Tube

The vortex generator – a replaceable, stationary component – controls air volume so that air flow and temperature ranges that the tube may generate can be modified.

1.4 Types of Vortex tube

Flow configuration, heating techniques (removal), and the organisation of low-pressure gas stream removal are the main technical and design features of Vortex tubes. A vortex counter-flow tube and a vortex parallel flow tube are the two most common types of cold exhaust tubes (uniflow). Non-cooled (adiabatic) and refrigerated (non-adiabatic) bay pipes are classified according to how they distribute heat (removal). Split vortex rods, on the other hand, are referred to as vortex tubes, self-evacuating vortex tubes, and vortex ejectors depending on how low pressure gas streams are evacuated. **[9]**

1.4.1Counter-Flow Vortex Tubes

According to the design of counterflow vortex tubes, the cold exhaust is located on the opposite side of the hot exhaust, as shown in Fig. 1.4. The working gas is introduced tangentially into the vortex tube via intake nozzles located close to the cold exhaust. As the gas moves through the tube, there is a powerful spinning motion created by the tube. The input gas is warmer on the outside of the stream than it is on the inside, and the gas cools down to the tube's centre as it passes through it. A part of the gas in the vortex tube reverses the axial component of velocity, causing the gas to transition from a hot to a cold state. The cold core gas is separated from the warm core gas by an aperture immediately below the flow entrance, and the pipe then leaves the left side. The warm peripheral flow exits the tube on the right side, where a valve is positioned to regulate the relative amounts of hot and cold gas in the tube.

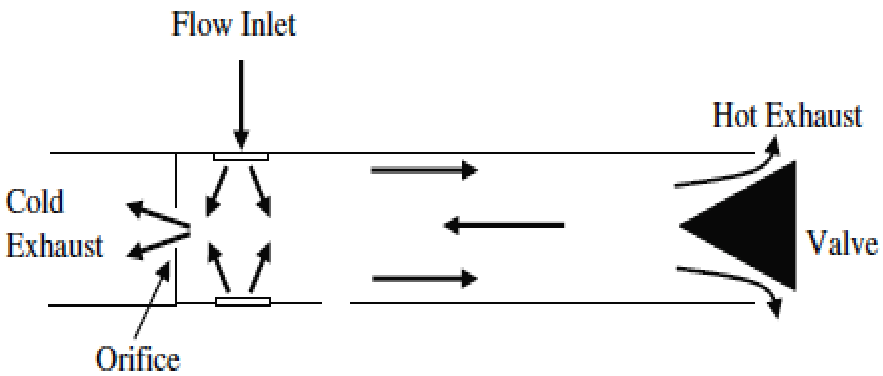


Figure 1.4: Counterflow Vortex Tube

1.4.2 Uniflow Vortex tube

In situations when cold exhaust is located on the same side as hot exhaust, uniflow tubes are utilised. This design is almost similar to the counterflow tube in terms of appearance. In addition to having a hole and a valve on one end of the tube, the other end is sealed near to its inlet paddle, which is one of its distinctive characteristics (Fig.1.5). In the opinion of many experts, uniflow tubes are less successful than counterflow tubes with proportionate counterflows in terms of performance. It is common practise to choose the counter-flow form as a consequence.

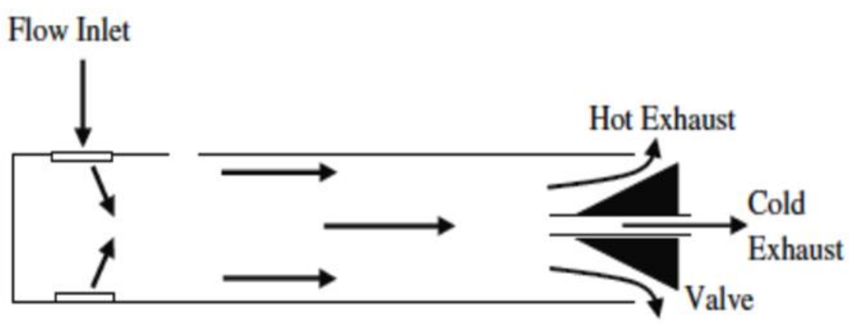


Figure 1.5: Uniflow Vortex Tube

1.4.3 Dividing Vortex Tubes

Figure illustrates a schematic splitting vortex tube. The most well-known and common form of RHVTs is the dividing vortex tube. The chilly and warm flows are similar. As illustrated in Fig. 1.6, it has up to 10 designs, and is utilised in different sectors.

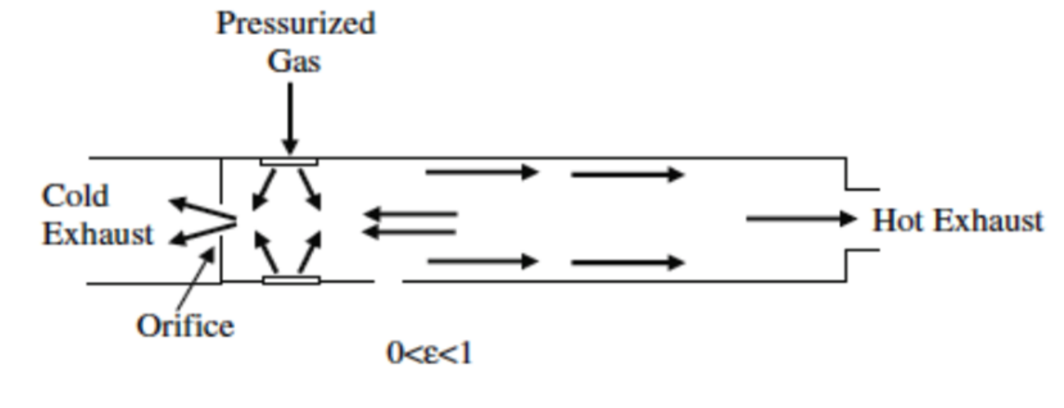


Figure 1.6: Dividing Vortex Tube

1.4.4 Uncooled (Adiabatic) Vortex Tubes

Heat transmission to the environment is ignored by adiabatic vortex tubes. It distinguishes from other vortex tubes; nevertheless, it is most effectively utilised in a combined regenerative throttling cycle with maximal cooling power at a high temperature level.

1.4.5 Cooled (Non-adiabatic) Vortex Tubes

Heat may be transferred via non-adiabatic vortex tubes from a thermal fluid to a coolant fluid. Another term for these tubes is "Cooled tube vortex." Figure 1.7 shows a cool vortex tube schematically. The cooled vortex tube varies from that of a split vortex tube because it is cooled down (20-30 kB) all gas that enters intake into the nozzle since the cooled vortex tube is closed, and all the gas entering intakes of the nozzle (20-30 kB) is cooled down, i.e. in such cases = 1. There is an insignificant cooling impact on the cooling vortex tube. It stands out from other vortex tubes, but is best used in a combined regenerative throttling cycle at a high temperature level with maximum cooling power.

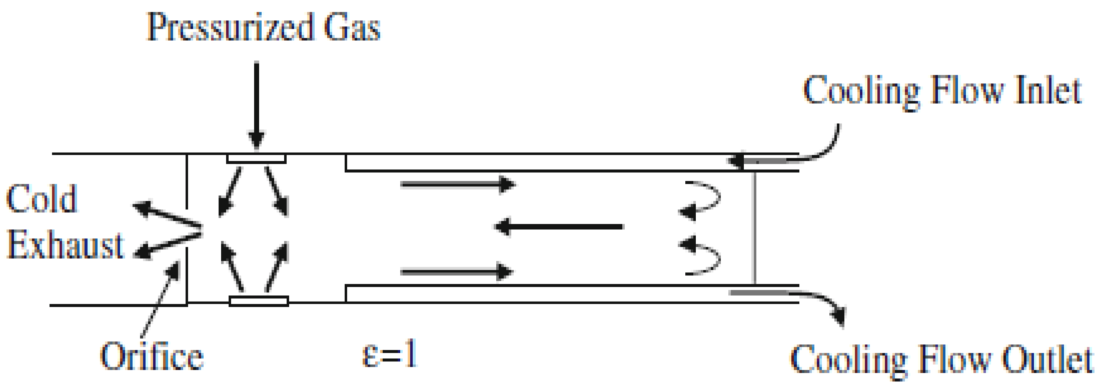


Figure 1.7: Cooled Vortex Tube

1.4.6 Dividing Vortex Tube with an Additional Stream

A dividing vortex tube configuration like the one illustrated in Fig. 1.8 is also a dividing vortex tube setup. The heat end of the control valve has an aperture in the centre that allows feedback gas to be injected into the vortex tube. Vortex tubes with dual circuits are another name for these tubes. The first circuit is a peripheral vortex circuit in which the working fluid enters the tube through the nozzles. The second circuit is an axial vortex circuit with a gas input through the vortex pipe at the hot end. If the excess gas has a low surplus pressure, the splitting vortex tube with additional stream ensures that 1 is delivered from a heat end to a reduced pressure zone on the axis. The splitting vortex tube with additional flow clearly demonstrates that an energy exchange may occur between the peripheral vortex and a non-twisted gas flow with a strong cooling effect: the adiabatic efficiency is 0.36 at 0.4 MPa outlet pressure. This design enhances the vortex tube's performance while also increasing the cooling power of the system.

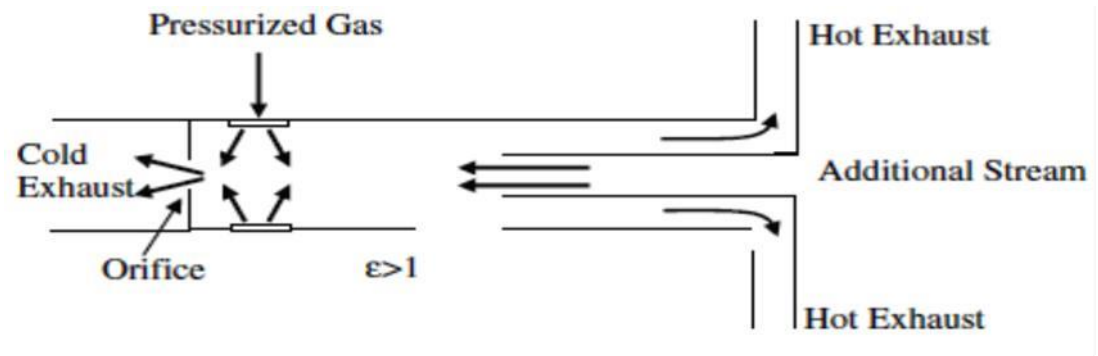


Fig 1.8 Dividing Vortex Tube with Additional Stream

1.4.7 Triple-Stream Vortex Tube

The three-stray vortex tube, unlike the typical dividing vortex tube, has an internal cylinder (tube) that produces an annular breach in the housing, into which the condensate released into the periphery is fed (Fig. 1.9). The amount of condensate instantly separated in the three-stream vortex tube may be as much as 40% to 90% of the total gas condensate. This indicates that the tube has a high efficiency as a separator.

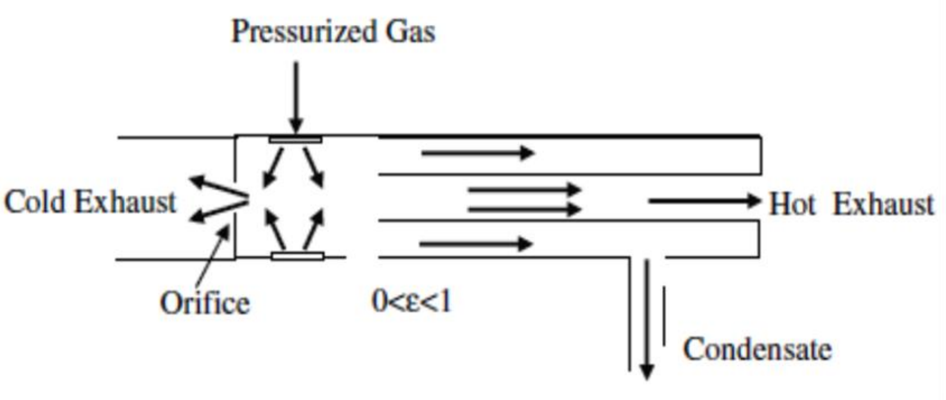


Fig 1.9 Triple-Stream Vortex Tube

1.4.8 Self-Evacuating Vortex Tube

By adding one or two non-twisted slot diffusers at the hot end of a pressure drop in the paraxial zone, the self-evacuating vortex tube has the highest refreshing effect (of all vortex tubes) (Fig.1.10). The self-evacuating whirlpool is intended for deep cooling of cylindrical masses and cannot transfer a cold gas stream to the surroundings.

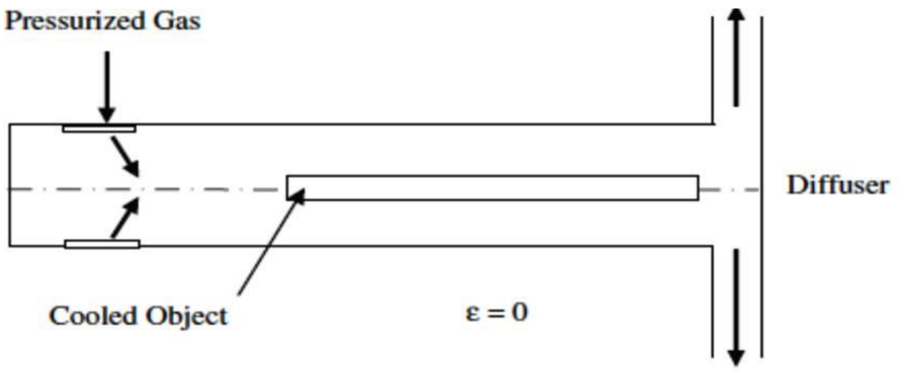


Fig 1.10 Self-Evacuating Vortex Tube

1.4.9 Vortex Ejectors

Due to particular geometric features and gas dynamic conditions, the vortex ejector may produce and maintain a pressure below the atmosphere in the paraxial region. The vortex ejectors (Fig. 1.11) differ from conventional direct throttle ejectors in that they are more smooth and efficient over a wide range of beginning settings.

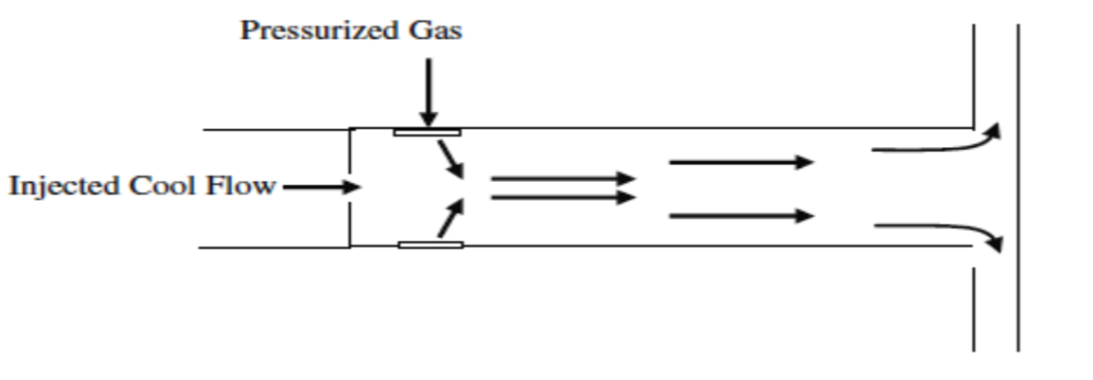


Fig 1.11 Vortex Ejector

1.5 Types and Number of Inlet Nozzles

Type, type and number of intake nozzles are very significant in vortex tubes. Several studies of various kinds and number of buckets have been conducted to get the optimal profile of the intake and the number of buckets. The intake nozzles have to be constructed to tangentially transform the flow into the vortex. All research showed. The majority of vortex chambers with a single circular inflow are round. The two chamber designs were developed by Martynovskii and Alekseev, including a chamber named 'Hilsch Whorl,' and the conclusion was that the circular chamber was more effective with two nozzles. Westley was employed in 1957 to enter a circular chamber with many rectangled nozzles.

The intake nozzle should have a slot form to it. Reynolds made a suggestion. Additional RHVT slots were suggested by Reynolds. He also said that adding more slots improves performance. Metenin used six tangential nozzles, and he created an archimedesian spiral in 1964, according to Fig. The vortex chamber and intake nozzle should be designed in such a way that the inlet is Archimedean in shape and the cross-section is slotted, according to Parulekar. Leites et al. used a single rectangular nozzle on their large industrial vortex tube. Saidi and Valipour tested three and four bowl vortex tubes in their experiments. The results showed that the nozzle has three intakes, which is better to the four intake nozzles in terms of cooling efficiency. Promvonge and Eiamsa-ard studied the energy and temperature separation of the vortex tube at a snail input experimentally. When compared to the original Tangential inlet pins, using the snail entrance may help to increase the cold air temperature drop and improve the vortex tube's efficiency. In a separate research, Promvonge and Eiamsa-ard discovered that "the number of intake nozzles increased the separation of the vortex tube."

CFD analysis, according to Behera et al., may be utilised to determine the best nozzle profile and number. There were two convergent nozzle numbers, a single helical circular bug, a single helical rectangular bug, a direct six nozzle number, and a convergent six nozzle number. In a 12 mm diameter vortex tube, they used five different nozzle configurations. The degree of swirl speed and radial fluctuating symmetry were used to assess dust performance. [9].

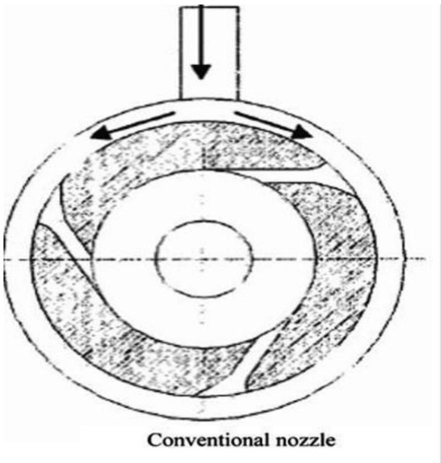


Figure 1.12: Conventional Nozzle

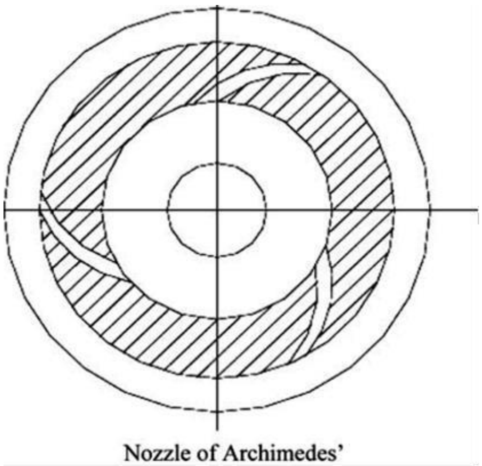


Fig 1.13 Nozzle of Archimedes

1.6. Advantages

Vortex tubes are simple, with no moving parts, without electricity or chemical compared to other refrigerant or heating devices, and have extended operating duration, and provide benefits. You just need to run compressed gas. Its main drawback is its poor thermal efficiency.

1.7 Objectives of the Study

- 3D model of the SolidWorks 2016 vortex tube.
- Fluent analysis of vortex tube on Ansys 16.0.
- Study on the insulated and non-insulated vortex tube behaviour
- Performance of vortex analysis improvement with various tube pressure, valve angles and pressures.
- We analyse energy balance, exergy balance, COP and energy efficiency with four different coolers - R1234 YF, R1234 ZE, R32, ISOBUTANE - with the use of cold exit temperature in the refrigeration system.

1.8. Organization of thesis

This research study is categorized into five segments are as follows: -

CHAPTER 1: Gives the general description of vortex tube, its type uses If various applications, pressure, design, materials used terminology, their classification, program definition and main purpose.

CHAPTER 2: This deals with a review of the past literatures on vortex tube analytical and experimental methods.

CHAPTER 3: It discusses research methods and outlines the thesis's issue. It discusses the many parameters involved, such as ANSYS modelling, building configuration, geometry, pressure used for analysis, and so on. The fundamental technique of CFD analysis.

CHAPTER 4: This includes a graphical representation of the findings as well as an explanation of the outcomes.

CHAPTER 5: It explains the results and concludes logically.

CHAPTER 2

LITERATURE REVIEW

2.1. Literature review

There is a lot of research on the energy separation phenomenon in a vortex tube. Instead, it offers a contemporary framework of the most essential elements, split into first and second-law thermodynamic analysis of the vortex tube, as well as experimental research to enhance the vortex tube's efficiency by altering its geometric and operational characteristics. The progress of CFD simulation of the complex inner flow of a vortex tube is next investigated. Following that, the findings of the experimental and numerical studies will be analysed in order to establish appropriate geometric parameters for the vortex tube under examination in this study.

Ahmad Al-Qaisia et. al. (2020) It was determined in this research that the thermal performance of the vortex tube could be improved by investigating the effects of changes in input pressure, conical valve opening percent, tube diameter, and hot gas gas tube length on the tube's thermal performance by using an experimental setup. Analytical data was gathered from four tubes with diameters ranging from 14, 21, 25, and 32 mm in the experiment, as well as four lengths of hot exhaust gas spanning 25, 50, 75, and 100 cm. The hot side length of 75 cm had the highest performance coefficient and the greatest cold temperature decrease, whether or not it was equipped with insulation. Furthermore, the thermal performance of the RHVT was shown to be optimal for internal diameters between 14 and 21mm. This implies that a thermal side length to inner diameter (Lh/D) ratio of 36 to 50 is optimal and highly recommended, which is in accordance with prior research and study findings.

Zangana et. al. (2020) The newly constructed Ranke-Hilsch vortex (RHVT) tube, such as a convergent-divergent-tube, is utilised in this research using experimental and computational fluid dynamic (CFD) techniques. The technique FLUENT is used to solve the control equation using three-dimensional compressible and turbulent models in the case of the conventional k-e turbulence model. The C++ is used for the FLUENT algorithm. Results of this research based on the CFD study results, show that the lowest temperature of reverse cold flux is not exactly located on the cold outlet of the vortex tube. This is consistent with the findings of the CFD study. At the end of this section, various suggestions and results are presented in order to obtain greater temperature separation in the vortex tube system. In addition, several further results in this article are shown by contrasting a convergent, divergent vortex tube with a basic vortex tube and examining the effect of this comparison on the maximum cooling point of the tube. In the present text, both the CFD and the experimental results are acknowledged. According to the data gathered, reducing the throat width from 8 to 2.5 increases the cooling power of the vortex tube.

Hong Yan et. al. (2020) The current study examines the temperature and the power separation of a novel converging vortex tube and the performance of a straight vortex tube. The temperature and energy of the convergent vortex pipe varied across all of the configurations examined and, compared to the typical straight vortex tube, certain variants exhibited superior operational performance. Temperature variation increased by 25° and performance coefficient by 32 percent (COP). This means that the cost of a vortex tube may be reduced and better performance still achieved. Convergent valve vortex tube work performance experiments. The experimented study was conducted using a number of convergent vortex tubes, including a heated tube diameter, tube length of a convergent tube and a vortex generator. The converging vortex tube may have much shorter hot tube, say 30 mm, in comparison to a 90 mm straight tube. The study has shown that the expense of a vortex tube may be substantially reduced if a conventional complicated hot and plug tube is substituted with a shorter, simpler converging tube.

T. Dutta et. al. (2020) For the same geometric and operational features, this article investigates In counterflow and uniflow vortex tubes, energy is separated. This is a test that is being carried out. To identify the cause of energy separation in different types of vortex pipes, a three-dimensional CFD research will be conducted. The flow field within the two pipes is studied, and the working and heat transfer rates between the core and the periphery are calculated. In counter and uniflow tubes, tangential shaving work is recognised as a source of energy separation in the periphery from the core zone, whereas heat transfer from the periphery to the axial region reduces energy separation.

Konstantin et. al. (2019) The bigger vortex tubes were investigated via the use of cyclonic extrusion of vortex chambers. The STAR-CCM+ CFD programme was used to simulate a conventional air-operated vortex tube counter-flux in a closed loop. The numerical approach was shown to be superior when compared to empirically accessible research from the literature. A design for a reasonably effective, high-capacity vortex tube that has been parametrically analysed. The setup, as well as different modifications, were numerically simulated under a set of defined external conditions. According to the results of the tests, most efficient was the chamber expansion of the intermediate vortex design, which was the most complex. This variant of the vortex tube was predicted to perforate by 15% better than the original device in a range of cold-flow fractions when compared to the original.

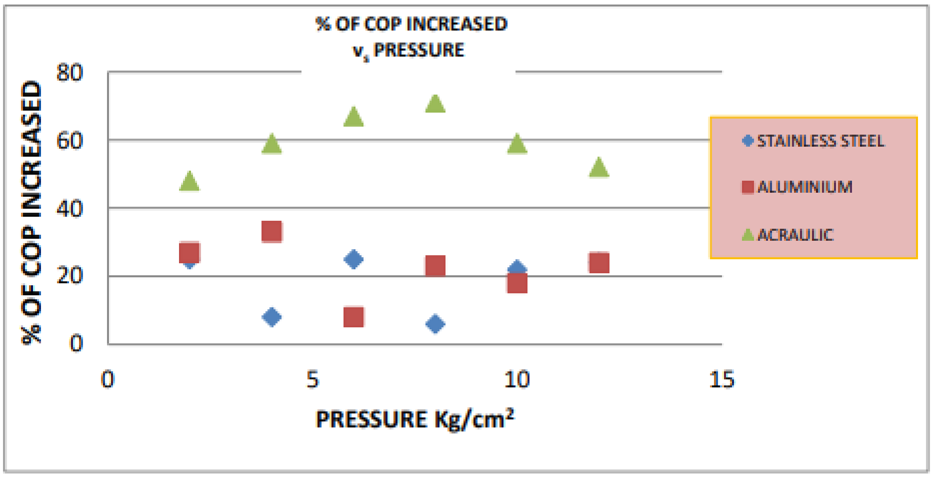
A. Aghagolia, et. al. (2019) This study develops a 3D model for the modelling of CO2 flow in a vortex tube and validates it using current experimental data. Hexahedral structured nodes generated in ANSYS meshing are used in the assumed k-turbulence model. Only 1/6 of the geometry is required due to the VT symmetry. In a parametric study, the validated CFD model is linked to the VT's thermodynamic model, with the intake pressure and cold weight percent chosen (550 kPa-to-1300 kPa) (0.2 to 0.9). Effects of VT energy and performance separation. For energy separation, the hot output and cool exit temperature discrepancies are addressed, both in terms of input VT temperature. Exergy measures, as well as cooling power, heating, and energy, are all part of the efficiency equation. With a constant input pressure of 1.300 kPa, the difference between the hot and cold exit temperatures varies from 10 to 78.9 degrees Celsius, while the cold exit temperature differential ranges from 44.2 to 9.7 degrees Celsius.

D.G. Akhmetov et. al. (2019) An experimental investigation of the flow structure of an incompressible fluid in a Ranque–Hilsch vortex pipe was carried out. We were able to measure the velocity distribution across the whole volume of the vortex tube by using a streamline pattern and laser Doppler anaemometry, which were both developed by the authors. The radial distributions in the Ranque-Hilsch vortex pipe were found to be distinct from those in a vortex tube with a single central exit hole and a tangential input hole, which was previously thought to be the case. To describe the physical mechanism of the temperature separation effect, a simple qualitative model based on the observed flow pattern may be utilised to explain the flow pattern. Comparing the RHVT to a vortex tube with a single central exit orifice, the azimuth velocity of the RHVT rises according to the increase in the radius of the tube at any axial position. An extremely simple and physically consistent qualitative temperature separation model, the ranque–Hilsch vortex tube may be built in a short period of time. It is based on the study of flow structure. Ranque's effect on toroidal volume is believed to be mostly due to this volume, which is the most significant factor.

Muhammad FadhliSuhaimi et. al. (2018) The effects of vortex tube length, cold flow diameter, and different working gases were investigated. There are three different tube sizes, three different cold exit diameters, and eight different gas types. For the numerical study of the effect, the free software Simflow was used. The results indicate that L=175 mm, d=4 mm, and helium are the best tube lengths, cold exit diameter, and working gas, respectively. To figure out how the vortex tube works, Researchers looked at its length, cold outlet diameter, and working gas. We used three different tube lengths, three different cold outlet diameters, and eight different work gases. The pressure is higher on the outside and in the centre. There is less pressure. This is due to the compressed gas in the centre, which causes the flow zone to be cool.

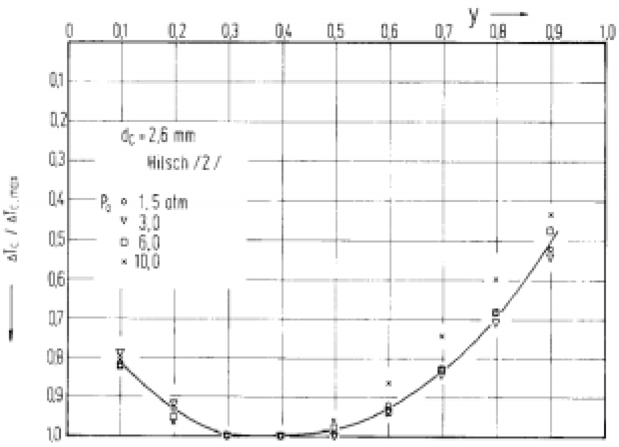
Maheswaran A et. al. (2018) The cooling effect of a vortex tube that uses air as the working liquid has been studied. Heat, pressure, discharge, mass flow rate, speed, and other flow characteristics were all measured. Our views have been tabled and collected. Pressures of 7-6 bar, 6-5 bar, 5-4 bar, and 3-4 bar were obtained as input pressures. The vortex tube is used as cooling equipment, spot coolers, and other applications in CNC machines. Because there are no moving parts in this technique, it requires very little maintenance. This research focuses on two aspects of the vortex tube involved: shape and thermophysical characteristics. It is obvious that the vortex tube's power is always proportional to the air intake. A tangential nozzle is difficult to position in a cylinder, and the Vortex generator can eliminate this difficulty. The vortex generator does not need a tangential nozzle. Vortex tube is ideal for a variety of applications because to its compactness, environmental friendliness, and lack of wear and tear.

K. Kiran Kumar Rao et.al. (2018) The cooling function is essential in human life since it has an impact on the human field: food grain and medication sustainability, massive stockpiles of chemical products and medicines, industrial cooling and air conditioning, and so on. Freon is a pollutant that depletes the ozone layer and poses health hazards to those who use cooling systems. Proxy cooling techniques are used to avoid such damage via extensive study. The vortex tube is a non-traditional freezing device that is easy to use and capable of separating hot and cold air from the necessary air supply without altering the field. Because of its small size, modesty, and efficiency, this gadget is useful for a variety of applications.



5 % of Cop Increased Vs Pressure by Stainless Steel, Aluminium alloy and Aeraulic VT.

SayaliDarekar et.al. (2018)A vortic tube is a simple device that converts a heated, non-chemical, or external energy flow from a hot and cold gas flow. There are no chemical reactions. This is a mechanical gadget with moving parts. The separation effect of temperatures describes the flow separation into low- and high-temperature regions. The vortex tube's performance is determined by two main variables. The first is an operational parameter like compressed air input pressure, and the second is a geometric parameter like nozzle count, nozzle diameter, cone valve angle, tube length, cold aperture diameter, and vortex material. The operational parameter is the first. The Vortex tube is a refrigerator that is utilised in industry and has unique functions".



Similarity relation, equation (25), compared with experimental data of Hilsch.

Mr. Shadab Shaikh Bismillah et.al. (2018)"As we all know, safety has grown into a critical component of business and helps to unite people. I believe that the ecological nature of research and development is the most significant aspect of today's research and development. The vortex tube is known for producing warm and cold air that is non-polluting to the environment when we utilise compressed air from a compressor. Tanks that are converted tangentially into vortex chambers The vortex tube separates the compressed air in the vortex tube into two parts, allowing the free vortex to emerge as the peripheral warm stream at the end of the conic wind and the forced vortex to emerge as the inner cold stream through the orifice. The peripheral warm stream at the end of the conic wind is known as the free vortex. While on the move, it may be utilised for a number of cooling purposes.

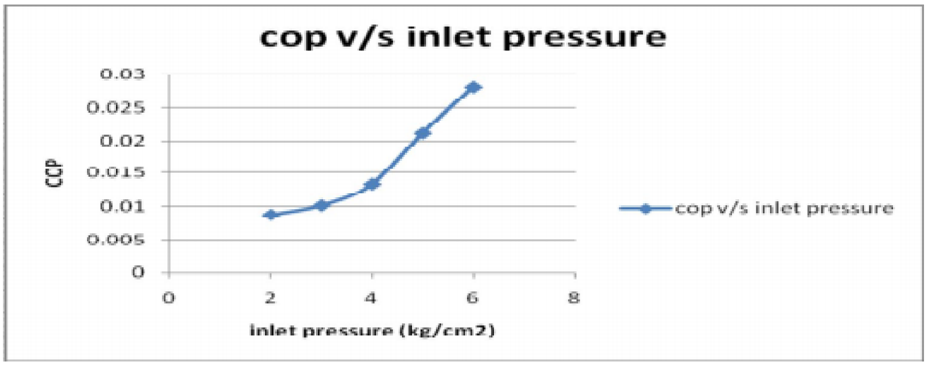
Maheswaran et. al. (2018) Rankquethe Vortex gas tube absorbs it and replenishes it, Hitsh. Significant expansion, centrifugal strength, secondary circulation, and friction are all factors that contribute to this reduction. For cooling and fluid operation, we designed an air vortex tube. Temperature, pressure, discharge, weight, speed, and other flux characteristics are all included in the measurements. Our thoughts have been collected and put on the table. 7-6 bar, 6-5 bar, 5-4 bar, and 3-4 bar were used as input pressures. Vortex tubes are utilised as cooling units in CNC, spot cooling, and other applications. Because there are no moving parts in this technique, it requires very little maintenance. The goal of this research is to focus on two geometrical characteristics and thermophysical variables in the vortex tube.

Deshmukh, et. al. (2018) This article uses exergy analysis to assess separation of energy and flow phenomena inside a vortex tube, changes in radial directions and maximum efficiency of the vortex tube, and evaluates energy separation and flow phenomena. Air is a fluid or coolant that works. During these operations multi-million dollars vortex tubes with different geometric characteristics were employed (number 1,2,3,4 Valve Angle 300,450,600,900). With a cold weight rate of between 0.1 and 0.9. These variables are used to assess the exergy. At 600 vane angles and 0.4022 in box 4 the maximum efficiency of the Exergy is achieved. Further study on the vortex tube is necessary to enhance exergy efficiency with various working fluids, L/D ratios, hole diameter, cross-sectional buckets and valve angle shape.

S. Karthik et. al. (2015) In this study, the flow rate calculations are measured, and the designs are compared to those developed by Hilsch, Reynold, and Albohrn in their previous work. In physics, a vortex tube is a system that generates both higher and lower temperatures at the same time at both ends of the tube. The use of PVC as a material yields results that are consistent with those obtained by pioneering scientists such as Hilsch and Reynold. The vortex tube's construction is such that it comprises of a hollow tube made up of metallic or fibre components, as well as an air flow control nozzle and a diaphragm or aperture. A spiral-shaped vortex is created by the airflow, which heats the air when it returns. When the air returns, it rapidly refreshes the air, resulting in a cooling effect as air travels through a tube diaphragm into the vortex tube.

Kiran D. Devade et. al. (2016) The effects of the cold end hole diameter, length by diameter ratio, and valve exit angles on the heating and cooling performance of a Ranque – Hilsch vortex tube with air as the working liquid are investigated in an experimental study. The tube and cold end apertures are made of brass in these experiments. There were four different L/D (15 single tube, 15–18 with 4° divergence angle), output valve angles (30°–90°), and cold end apertures (5, 6 and 7 mm) produced. The exergy loss and efficiency of exergy were measured after increasing the entry pressure to 200 kPa, With the cold-end aperture width increasing, the loss of exergy between the cold and hot fluid is reduced, according to the results of the experiment. Exergy efficiency decreases as the L/D ratio rises. Furthermore, the different vortex tube is found to produce less exergy than the simple tube. The loss of vortex tube thermal exergy is influenced by valve angles.

Anil Kumar Bodukuri (2017) It is important to note that the Vortex tube is a mechanical device with no moving components. As a consequence of the air transmission, two air streams with very high temperatures are generated. Due to its unique characteristics, the vortex tube is used for a variety of applications, including heating and cooling. The vortex tube generates hot and cold air at the same time at both ends of the tube. Low temperatures are the subject of this article, since they are what causes the cooling effect. There is a comparison of the number of nozzles, apertures, and companies produced in four different tubes.



COP Vs Inlet Pressure.

T. Dutta et. al. (2020) This research compares energy separation in counterflow and uniflow vortex tubes that have the same geometrical and operational properties. In this kind of vortex-tube, a 3-D CFD research is also carried out to determine the cause of the energy separation disparity. Inside both pipes, a flow field is examined, and work and heat transfer rates between the centre and peripheral areas are calculated. Energy is separated from the core region to the periphery due to tangential shear work, while heat transfer from the peripheral to the axial zone reduces energy separation between counterflow and uniflow vortex tubes. In a uniflow vortex pipe, however, the transfer of considerably greater sensible heat from the perimeter to the core region causes energy separation to be decreased further by reducing the heat output and increasing the cold output temperature.

Seyed Ehsan Rafiee et. al. (2013) The convergence ratio is the primary focus of the experimental study, which examines the inlet, nozzle, and number of intakes in various configurations and conditions. When a nozzle convergence ratio of 1e2.85 is used, the impact of the nozzle convergence ratio is studied. Using computational fluid dynamics (CFD) to develop a tool that can be used anonymously in a variety of operating conditions and geometries, the researchers hope to develop an extremely useful resource for optimising the design of vortex hoses and testing their suitability for use in new applications. For the purpose of estimating the performance of the vortex tube system, a computational fluid dynamics model was employed. The numerical investigation was carried out with the help of a complete 3D (3D) steady state CFD simulation carried out using FLUENT 6.3.26. It is necessary to solve the flow equations using the turbulence model. The findings of the simulation were confirmed via experiments, which were also carried out. In this instance, the first objective of numerical research was to validate these findings with experimental data, and the second goal was to optimise the experimental model in order to get the best results.

A. S. Deshmukh et. al. (2018) The main objective is to use an exergy analysis to study energy separation, flux fluctuation in the radial direction, and the most efficient effect of the vortex tube in order to investigate energy separation and flow phenomena in a vortex tube. Air is considered a working fluid or coolant. Multi-nozzle vortex tube with various geometric characteristics (no no nozzles 1,2,3,4, valve angle:300,450,600,900). In cold mass, it ranges from 0.1 to 0.9. Exergy research makes use of these variables. The Exergy's maximum efficiency is reached at 600 valves and 0.4022 at nozzle 4. The cold mass percentage of the vortex tube improved Exergy Efficiency. The maximum CMF of 0.9 has been discovered. It does a good job of matching the previous material. With L/D 22.5, the intake nozzle's efficiency improves. It reaches its maximum at nozzle 4. The angle of the conical valve has a considerable effect on exergy efficiency. The valve angle 600 has the highest efficiency.

R Liew et. al. (2012) Temperatures, pressures, and speeds were measured using a Ranque-Hilsch vortex tube. The results show that the cooling power is higher than the heating power due to heat loss in the surroundings. This heat loss becomes the most significant thermodynamic process in large cold fractions. Laser Doppler anemometry was used to obtain the speeds. A non-intrusive measured methodology shows the three-dimensional velocities of gas and their standard anomalies in the vortex tube using this method. The isotropic turbulence in the core region of the vortex tube is shown by turbulent fluctuations, which are defined by standard variations. The results show that the cold output temperature is lowest when output pressures are low and mass flow is high. Due to the high temperature of the gas near the RHVT's walls, the temperature for hot escape increases with ", and heat losses occur in the region. Because of the heat loss, the cooling capacity exceeds the heating capacity. The isentropic efficiency of the RHVT as a heating device, however, is higher due to the high heat exit pressure.

K. Dincer et. al. (2011) It has been decided to use three Ranque-Hilsch vortex tubes, each having an inner diameter of 9 mm and a length/diameter ratio of 15 for this experiment. A traditional RHVT cascade and another hot cascade from RHVT were used to compare and assess their performance. The difference in temperature between the heating supply and the intake was used to determine the performance of the system (DThot.). The DThot values for Ranque-Hilsch vortex tubes, which were determined experimentally, were found to be higher than the DThot values for classical Ranque-Hilsch vortex tubes. It was determined that the total input exergy, total output exergy, total loss of exergy, and the hot stream efficiency were all greater than zero using experimental data. Both the conventional RHVT and RHVT hot waterfalls demonstrated that the total loss of exergy decreases as the percentage of cool flow increases in both experiments. Furthermore, it has been shown that the RHVT type of heat cascada is more energy efficient than the conventional RHVT. The steaming waterfall The Ranque-Hilsch vortex tube has a high hot waterfall value, which results in high hot exergy efficiency when used.

M. Yilmaz et. al. (2009) Several recent papers were examined, and complete information on design requirements for vortex tubes was provided as part of this investigation. The Vortex's tubes were classified, and the types of tubes were defined. The results of previous tests, as well as theoretical predictions, have all been utilised to define all of the criteria for the construction of vortex tubes. Finally, the design criteria are outlined in detail.

Upendra Behera et. al. (2005) The optimization of the Ranque – Hilsch vortex tubes is accomplished via the use of CFD and investigational investigations. Using CFD, the number of dust nozzles and various kinds of dust profiles are examined. The components of swirl speed, axial speed, and radial speed, as well as secondary flow patterns, were all examined in this study. Optimal cold end diameter (dc), length-to-diameter ratios (L/D), and optimal parameters for the highest hot gas temperature and the lowest cold gas temperature are determined and experimentally verified using computational fluid dynamics (CFD). The performance coefficient (COP) of the vortex tube was determined in the context of both a heat engine and a refrigerator.

Kiran D. Devade et. al. (2016) The counter-flow heating and cooling function of the Ranque-Hilsch vortex tube, which works with air as a liquid, is affected by the cold end of the aperture diameters, the distance between the diameters, and the valve output angles. In these experiments, the tubes and cold end apertures are constructed of metallic material. Experiments have shown that increasing the diameter of the cold tip significantly decreases the exergy loss between the heating and cooling fluids. The energy efficiency of the system diminishes as the L/D ratio rises. There's also evidence that a differentiated vortex tube loses less energy than a single tube. At the heat end, the angle of the valves affects the loss of vortex tube exergy.

K. Dincer et. al. (2011) Three Ranque-Hilsch vortex tubes with a 9mm internal diameter and a length-to-diameter ratio of 15 were used in this study. They were divided into two types: hot waterfall RHVT and traditional RHVT. The temperature difference between the heat and the intake makes conducting a performance analysis difficult (D Thot.). The Ranque-Hilsch vortex tubes' D Thot values were greater than the D Thot values of standard RHVT. Overall input exergies, entire output exergy, total loss of exergy, and hot stream efficacy were determined using experimental data. We found that total energy loss as a proportion of cold flow was decreased in both the conventional RHVT and the RHVT hot waterfall. Furthermore, heat-cascade RHVT has been shown to be more efficient than traditional RHVT.

Seyed Ehsan Rafiee et. al. (2013) Dust, input pressure, and the number of intakes convergence are all investigated experimentally. The effect of nozzle convergence ratios in the 1e2.85 range is investigated. The research's primary goal is to demonstrate how CFD can be utilised to create a tool that is reliable in a wide range of operating circumstances and geometries, providing a strong tool for optimising vortex tube design and evaluating its usage in new applications. A computational dynamics model was developed to estimate the performance of the vortex tube system. Numerical study used FLUENT 6.3.26 to do a full CFD (3D) simulation. The Key-3 turbulence model is used to solve flow equations in this model. There have also been tests to confirm the simulation's results. The initial goal of numerical research was to verify these conclusions using experimental data, and the second goal was to optimise the test model to get the best possible results.

2.2. Conclusion of Literature Review

The intake pressure is the required power unit for energy separation into a vortex tube. The temperature difference between the output streams increases as the input pressure increases. The vortex tube's effectiveness varies with temperature. In order to retrieve parameters, the energy separation and separation of flow efficiencies are adapted for measuring the unique features of the vortex tube. The magnitude of the opening however doesn't increase further, because the length of the vortex tube rises but only up to the critical length. As the diameter of the vortex tube increases, then the magnitude of angular velocities decreases.

CHAPTER 3

RESEARCH METHODOLOGY

3.1. Research Methodology of CFD analysis

The answer to the flow problem is stated at nodes in each cell. The cell number in the grids determines the precision of the CFD result; the higher the cell number, the more accurate the result. The accuracy of the solution, as well as the related load, such as the need for computer hardware and calculation time, are all influenced by the fine grid quality. The majority of constructive mesh is usually uniform: finer in areas where there are more variations between neighbours, and coarser in areas where there are only small changes. The ability of the CFD user determines the accuracy and cost of the solution.

Domain geometry and grid design is defined in CFD projects about 50 percent of the time. All main programmes include a CAD interface which allows the importing of data from the proprietary surface models and mesh generators such as SolidWorks to increase the output of CFD staff. Until now the pre-processor additionally allows the user to admit normal fluid material characteristics and to request specific physical and chemical designs in addition to the main fluid flow equation.

3.2. Boundary Conditions & Physical Parameters

The boundary conditions are as follows:

- Stagnation limits are provided for admission at a total pressure of 0.5422MPa (6 Bar) and a total temperature of 298.
- Cold exit II and static input pressures have been calculated according to experimental measurements.
- The hot outlet and the cold outlet are the pressure outlet limit. The static heat output and cold output-I have been amended such that the fractions of cold weight change. Hot pressure outlet and cold I pressure outlet are retained

above cold outlet II to reduce flow via hot and cold I outlets. zero hot and hot temperature gradient; no slip and adiabatic wall circumstances.

3.3. Physical Parameters of the model:

The present CFD model was utilised to collect all experimental data. All geometric characteristics of the Bramo model are maintained constant in addition to the Bramo model for the investigation of length effects on the performance of vortex tubes. Four additional versions with various lengths were produced by changing the length of the model. The performance of turbot pipes of 11 - 40 diameter and six straight nozzles was thus investigated. This is the reason for this. The four L/D models of 11, 21.3, 32.2 and 40.7 are investigated with their fixed diameters. The dust is set at 3 mm in diameter. Cold outlet I and hot outlets are axial holes with a diameter of 6 mm and 9 mm (Opening area is partially closed by conical controlled plugs to reduce the flow through). Cold flow II is set at 3 mm in diameter (Area is 7.0685mm^2). Vortex Tube has six nozzle numbers for research and different L/D ratios.

3.4. Thermo-physical properties of working medium

“Working medium	: Air
Density	: Ideal Gas equation
Specific Heat	: 1006.43J/Kg-K

Thermal Conductivity: 0.0242W/m-K”

3.5. Solver

3.5.1 Governing equations

“Applying the conventional K- β turbulence model Bramo et al..[109] to the compressible turbulent high whirling current inside the vortex tube, the 3-D stability status is computed and therefore may simulate a turbulence effect. The conservation of mass, energy and momentum thus equalled the relationships”:

-Continuity Equation

“Rate of increase of mass in fluid element = Net rate of flow of mass into fluid element”.

$$\frac{\partial}{\partial x_j}(\rho u_j) = 0 \quad (3.1)$$

Momentum Equation

Newton's second law states that fluid particle dynamic change rates are the total particle strength.

$$\frac{\partial}{\partial x_j}(\rho u_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}[t_{ij} + \tau_{ij}]$$

Where P is the pressure and turbulent flow pressure (Reynolds) according to these definitions:

$$t_{ij} = \mu\left[\left(\frac{\partial_{u_i}}{\partial x_j} + \frac{\partial_{u_j}}{\partial x_i} - \frac{2}{3}\delta_{ij}\frac{\partial_{uk}}{\partial x_k}\right)\right]$$

“Where is the dynamic viscosity of the fluid and where is the draic delta, = 0 for I = 1 for I = J and turbulent shear loads”:

“Hypothesis Boussines q is a popular approach to represent the stresses of Reynolds and is provided as follows”:

$$\tau_{ij} = \mu_t\left(\frac{\partial_{u_i}}{\partial x_j} + \frac{\partial_{u_j}}{\partial x_i}\right) - \frac{2}{3}\left(\rho k + \mu_t\frac{\partial_{u_k}}{\partial x_k}\right)\delta_{ij}$$

“The Kronecker Delta and the K are the cinematic energy of where dynamic [N.s.m-2] is, is turbulent viscosityturbulence”.

$$\frac{\partial}{\partial x_j}(\rho u_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial_{u_i}}{\partial x_j} + \frac{\partial_{u_j}}{\partial x_i} - \frac{2}{3}\delta_{ij}\frac{\partial_{uk}}{\partial x_k}\right)\right] + \frac{\partial}{\partial x_j}(\tau_{ij}) \quad (3.2)$$

The term of τ_{ij} is also represented as $-\rho u_i' u_j'$, [ms-1] and u" are variable components of velocity, where u stands for average mass velocity, [ms^{-1}].

-Energy Equation

“The energy equation is derivative from the first thermodynamic law stating that the fluid particle's energy shift rate is equal to the heat rate plus its working rate”.

$$\frac{\partial}{\partial x_i}\left[u_i\rho\left(h + \frac{1}{2}u_j u_j\right)\right] = \frac{\partial}{\partial x_j}\left[k_{eff}\frac{\partial T}{\partial x_j} + u_i(\tau_{ij})_{eff}\right],\ k_{eff} = k + \frac{C_p\mu_t}{P_{r_t}} \quad (3.3)$$

$$\tau_{eff} = \mu_{eff}\left(\frac{\partial_{u_j}}{\partial x_i} + \frac{\partial_{u_i}}{\partial x_j}\right) - \frac{2}{3}\mu_{eff}\frac{\partial_{u_k}}{\partial x_k}\delta_{ij}$$

$$\mu_{eff} = \mu_t + \mu$$

Here μ is Dynamic viscosity [$N.s.m^{-2}$] and μ_t is turbulent viscosity [$N.s.m^{-2}$].

If the work fluid is considered an optimal gas, the compressibility result must be compulsory:

$$P=\rho RT \qquad (3.4)$$

The turbulent viscosity, μ_t, is enumerated as:

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

The kinetic turbulence (k) and the dissipation rate (μ) of the equations may be calculated:

$$\frac{\partial}{\partial x_j}(\rho k u_j) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + \tau_{ij}\frac{\partial u_i}{\partial x_j} - \rho\varepsilon + \rho G \qquad (3.5)$$

$$\frac{\partial}{\partial x_j}(\rho\varepsilon u_j) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + C_{\varepsilon1} f_{\varepsilon1}\tau_{ij}\frac{\partial u_i}{\partial x_j}\frac{\varepsilon}{k} - f_{\varepsilon2} C_{\varepsilon2}\rho\frac{\varepsilon^2}{k} + \rho E$$

"Where, (σ_ε ,$C_{\varepsilon1}$,$C_{\varepsilon2}$) are the model constants and ($f_{\varepsilon1}$, $f_{\varepsilon2}$) are the damping functions, and E is the additional term. G is the additional turbulence term and, σ_k and $\sigma_\varepsilon$are the turbulent Prandtl numbers (Pr) for *k* and *ε"*.

"The models are given as follows for K- turbulence model, damping functions and extra terms".

$\lambda = 0.11$, $\sigma_\varepsilon = 1.3$, $\sigma_k = 1.1$, $\sigma_\varepsilon = 1.3$, $C_{\varepsilon1} = 1.44$, $C_{\varepsilon2} = 1.92$.

$$\beta_\mu = \exp\left[\frac{-3.4}{\left(1+\frac{R_t}{50}\right)^2}\right]$$

$$f_{\varepsilon1}=1.1,\ f_{\varepsilon2} = 1-0.3\exp(R_t^2)$$

$$G = -2\frac{\mu}{\rho}\left(\frac{\partial\sqrt{k}}{\partial x_i}\right)^2$$

$$E = 2\frac{\mu}{\rho}\frac{\mu_t}{\rho}\left(\frac{\partial^2 u_i}{\partial x_j \partial x_k}\right)^2$$

$$R_t = \frac{\partial k^2}{\mu\varepsilon}$$

The Navier Stokes equations (compressible) are coupled with the regulating equations using rapid digital methods for the energy equation in order to produce a trend of fluid production and transfer of the temperature of the vortex tube. The data from the tests on pressure and temperature are provided as an entrance for the exam.

The governing equations will be resolved many times to fulfil the specified limit conditions of the CFD computing domain. Each error or residual is documented for the general preservation of the governing equations. The residual is set to 1 x 10-10, which indicates that the solution converges on an error equal to or less than 1 x 10-10 (or the governing equation is preserved). The first simulation of 520 iterations is followed by non-convergence of the solution (until the mass flow maximises). Diverse factors affect, in combination with the size and size of the final residues, how close to the completed CFD solution is to the precision. After 520 iterations (1 x 10-3), the remainder were still high and indicated that more iterationwere necessary for the correct solution. The following simulation was carried out for up to 600 iterations and showed virtually no residue reduction.

Grid Independence Test:

The present approach uses a mesh of unstructured inflation (boundary layer mesh). The cold air temperature was used as a criterion to check the grid sensitivity of computed results. The count of cells ranges from rough to fine, between 60 000 and 200 000. Following an increase in calculation domain fineness, cold temperatures have reduced to an optimum value of 148,700 cells. All computations were then made using a 148700 cell domain.

Smaller cell sizes offer more performance, but solving the system takes longer. Different cells are therefore taken into account for diverse regions. For example, the inlet and outlet crossing requires precise dimensions, while the rest of the tube is suggested to measure size.

Post Processing of Results

The numerical conclusions of the preliminary CFD postprocessing will be evaluated and investigated in this section. The numerical model data will then be processed to display the information interactively under the form of contour tracks (temperature variables), x-y graphs and other instruments (variable values in the domain). This section examines the post-processing results of the temperature transmission.

The objective of this numerical research is to compare and assess the acceptability of the numerical findings subjectively. Compared to these numerical findings, x-y-plots are better. The rotational and radial pressure profiles are presented at several measurement stations at different position Z/L where Z is the Z-Coal (tube axis) and L is the tube length.

In the final evaluation of stalls, cabinets, and bridges, the FEM is employed to restrict its use in the field of structural construction. The goal of dynamic structure research is to find regular frequencies, modes, and structural reactions. The design also shows its structures, such as atomic vessels, the control structure, and the reactor segment's dynamic reaction, using limited component technological ideas. Furthermore, even the biological structure employs a minimal component approach to conduct skull study. Exhumation, sub-ground, and advanced study of its component technique may be linking the dam supply system's underlying geomechanics.

The component strategy statement fulfils the requirement to assess current complex structures and structures, usually with no formal preparations for the management of equilibrium circumstances.

3.6. CFD SOFTWARE – ANSYS

ANSYS was developed by Dr. John Swanson. One of the FEA's pioneers was the idea of a PC-like building, which Inc. had hoped to market in 1970. ANSYS Inc. supports and provides comprehensive, adaptable design frameworks that enable companies to manage a wide range of research issues in addition to expanding their existing programming and technological interests. ANSYS Inc. is the industry's forerunner. That is their current situation. It also offers a methodical approach to construction and installation, allowing customers to avoid costly, time-consuming construction and interruption cycles. Customer usability, information similarity, multi-stage help, and multilateral scientific combination capabilities are all features of the ANSYS examination and reproduction equipment.

3.7. Evolution of ANSYS Program

ANSYS is now a multipurpose strategic research programme which has considered various strengths around the globe. The software is very creative and easy to use. Each download includes new and updated features to increase the flexibility, utility and speed of the programme. ANSYS allows specialists in these areas to comply with current criteria and standards.

3.8. Outline of the program

The package ensures an accurate, lasting and integrated design assessment. Software runs from PCs through workstations and supercomputers on large computers and systems. ANSYS offers file consistency across platforms and families of applications. With structural information and repaired work, clients may load plan templates supported by a PC to ANSYS. This allows all ANSYS users to respond to a wide, adaptable design.

3.9. Methodologyand Modelling

This section seeks to optimise air conditioning analysis using ANSYS-described computer fluid dynamics. Different steps are explained in detail for the process. Flow simulation is a sophisticated method, integrating fluid dynamics with turbulent movement, on internal combustion engines. Either a physical fluid with flow questions that is solvent experimentally or numerically. The numerical simulation is better suited for parametric research and has an increased effect on every mobile fluid domain by using fastening control equations. In recent years, the field of mathematical techniques has developed wonderfully and has a significant impact on the assessment and reaction to complex flux issues. Computer Fluid Dynamics (CFD) is an important tool in almost every fluid-dynamic area from mathematical curiosity. CFD is regarded as a relation between theoretical dynamics of pure experimental fluid and natural fluid. Until these days researchers had to understand complicated wafting problems on the basis of time-consuming and costly experiments. The breadth of the experiments needed to read complicated engineering problems is significantly reduced as digital computers and digital simulation techniques developed.

3.10. Computational Fluid Dynamics

By simulating the many fluid-flowing events on the computers, CFD offers solutions for fluid flow equations. This simplifies the work and duration required in order to minimise many engineering additives design configuration. These pictures complete a 3-dimensional numerical simulation to assess the effect of a 4-hr DI diesel engine with multiple brass chamber geometers on the cylindrical waft system, using open CFD software.

3.11. Basic Structure of A CFD Code

“Each novel fluid glide scenario may require time and impertinence to develop a CFD code. This has led to the development of industry standard CFD codes, including KIVA, AVL FIRE, STAR-CD, Fluent, ANSYS CFD and Open Foam. The GUIs (graphic user interfaces) allow the client to utilise an optimised code that includes the connection of trouble fixation devices to deal with fluid glide, heat transfer problems, etc. With three levels of fluid creation, the usage of the conventional CFD code is complicated.”.

1. “Pre-Processor
2. Solver
3. Post-Processor”

3.11.1. Pre-Processor

The pre-processing stage begins with a geometric assessment of the fluid flow issue. It can finish a geometry-assisted schematics that has been verified (CAD). When the geometry is constructed, an appropriate mesh may be generated. It may be described as a mesh that allows for the separation of smaller volumes or components. When a realistic mesh is constructed, density, viscosity, and turbulent flow are also known. The cell faces, including temperature, tension, and velocities, may be restricted by circumstances as part of a stream along an entrance, exit, or wall. Because the practical response is dependent in part on mesh quality, pre-processing is the most essential step in creating a solution.

3.12.3. Post-Processor

This kind of stress and flow should be developed for layout analysis utilising secondary factors. Most industrial CFD programmes offer an after-processing personal computer to calculate the secondary variables and the supply range for the graphs using nodal data from the simulation (contour in addition to line diagrams). This includes:

- Domain geometry and grid show
- Vector plots

- Line and shaded contour plots
- 2D and 3D surface plots
- View management and colour postscript output

These applications provide animation to show dynamic results. All programmes generate, in addition to visual, reliable alphanumerical output and export documents for processing outside the code.

3.13. ANSYS CFD (Fluid flow) Workbench Environment

“The ANSYS CFD (Fluid Flow) programme is completely integrated into the ANSYS workbench environment, the basis for the whole spectrum of engineering simulation solutions. Its flexible architecture enables users to easily install anything with basic and sophisticated interactive structures from generic fluid glide analysis. The CFD fluid waft equipment with an ANSYS is schematically shown in Figure 4.1. In many design factors the overall performance may be easily monitored or many alternative designs evaluated. In the ANSYS workbench environment for common equipment such as geometry and meshing instruments, packages from a variety of simulation disciplines may be utilised”.

Heat Transfer and Radiation: In addition to managing the convective electricity supply with the fluid float, ANSYS CFXD software offers a combined heat exchanger (CHT) capability for solving thermal conductivity in solids. There are a broad variety of models for capturing various types of radiation heat, either totally or semi-transparently between and in fluids and solids.

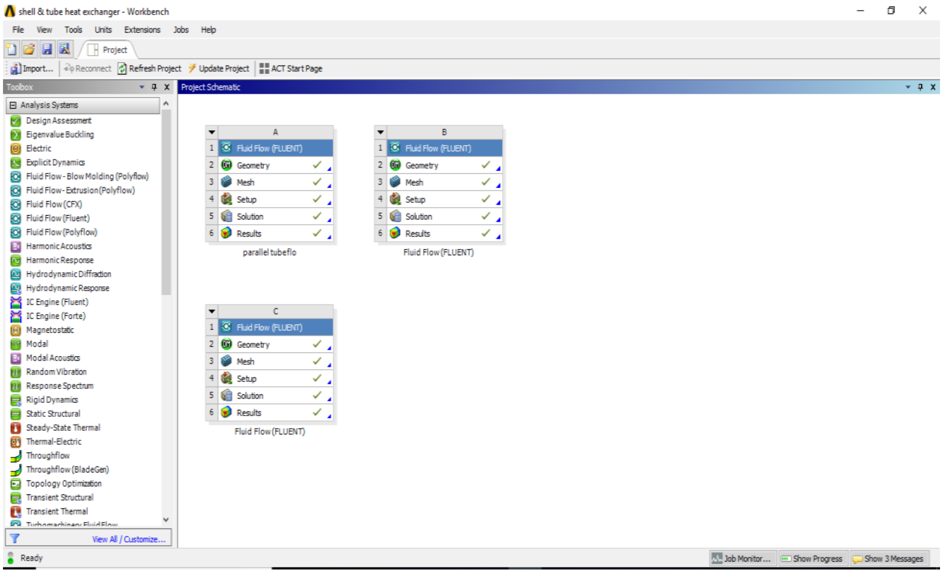


Figure 3.1: ANSYS workbench project schematic with fluid flow (CFD)

“In CFD simulation, the fluid-analysis tool inside the Project Schematic (Workbench) Fig:

- Building a mesh for geometry
- Building a mesh for geometry
- Create or import a geometry
- Build a mesh.
- Build an estimate a suitable method to ship to the solver control
- show the solver to get a system.”.

3.14. Create a Geometry

“All engineering simulations begin with geometry, which represents the importance of the layout for a structural assessment or the amount of air in fluid or electromagnetic field. The engineer has either CAD geometry or creating geometry from scratch (a laptop-assisted design).

3.14.1. SolidWorks

SolidWorks was founded in the late 1800s by Jon Hirschtick, who used the $1 million he won as a member of the MIT Blackjack Team to start the business. After that, a team of designers and engineers is put together to develop 3-dimensional CAD software. It was created to be low-cost, easy-to-use, and compatible with Windows. SolidWorks was launched in November 1995. It was the first significant Windows modeler. This is seen as a significant step forward in CAD development. Despite the fact that AutoCAD was launched first, SolidWorks brings something unique to the table in terms of 3D modeling. As a result, 3D CAD became a defining feature of the 1990s. SolidWorks altered the way engineers came to life throughout the months.[12]

3.14.2. SolidWorks Features:

In addition to AutoCAD blocks, SolidWorks software allows imports of DWG and DXF data that allow 3D models from 2D data to be developed. SolidWorks has a variety of cute applications for designing and making 3D designs quicker, such as Design Clipart which allows you to drop and move DWG file views to SolidWorks software models, and View Folding which allows the view of imported 2D drawings to be monitored and modified in order to simplify the creation of a 3D model. SolidWorks can also import AutoCAD 2D blocks to create more 3D capabilities in the programme. SolidWorks is renowned for the ability to generate accurate 2D drawings for 3D CAD. Often 2D drawings and sketch models are produced using SolidWork. Users will then extrude their creations into 3D using a variety of tools and software. More than simply a basic drawing platform, SolidWorks is aiming towards. Methods of simulation have been included, enabling designers, as stated above, to evaluate their components using actual world circumstances.[13]Simulation and modeling of heat transfer systems and functions, fluid streaming and fluid motions and life cycle (LCA) systems and mathematical (computation) methods for fluid dynamics include (CFD). The programme has further advances in rendering capabilities that enable users to see exactly what components they design.12 Other SolidWorks tools, like PDM and a range of electrical applications, make it much simpler to create schemes and circuit data appropriately. It also provides advanced rendering capabilities. It contributes to the knowledge that SolidWorks offers its own set of difficulties to its designers and consumers from a wide variety of industries. This will first impact a number of industries dependent on SolidWorks for daily operations.[13]

3.14.3. Dimensions of Vortex Tube

Table 3.1: Dimensions of model

Sr. No.	Element	Dimensions (mm)
1	Tube diameter	8
2	Nozzle diameter	1.5
3	Tube diameter	8

4	Tube length	80 ,112, 144
	Insulation wood	2 mm thickness
5	Cold exit diameter	3
6	Hot exit diameter	8

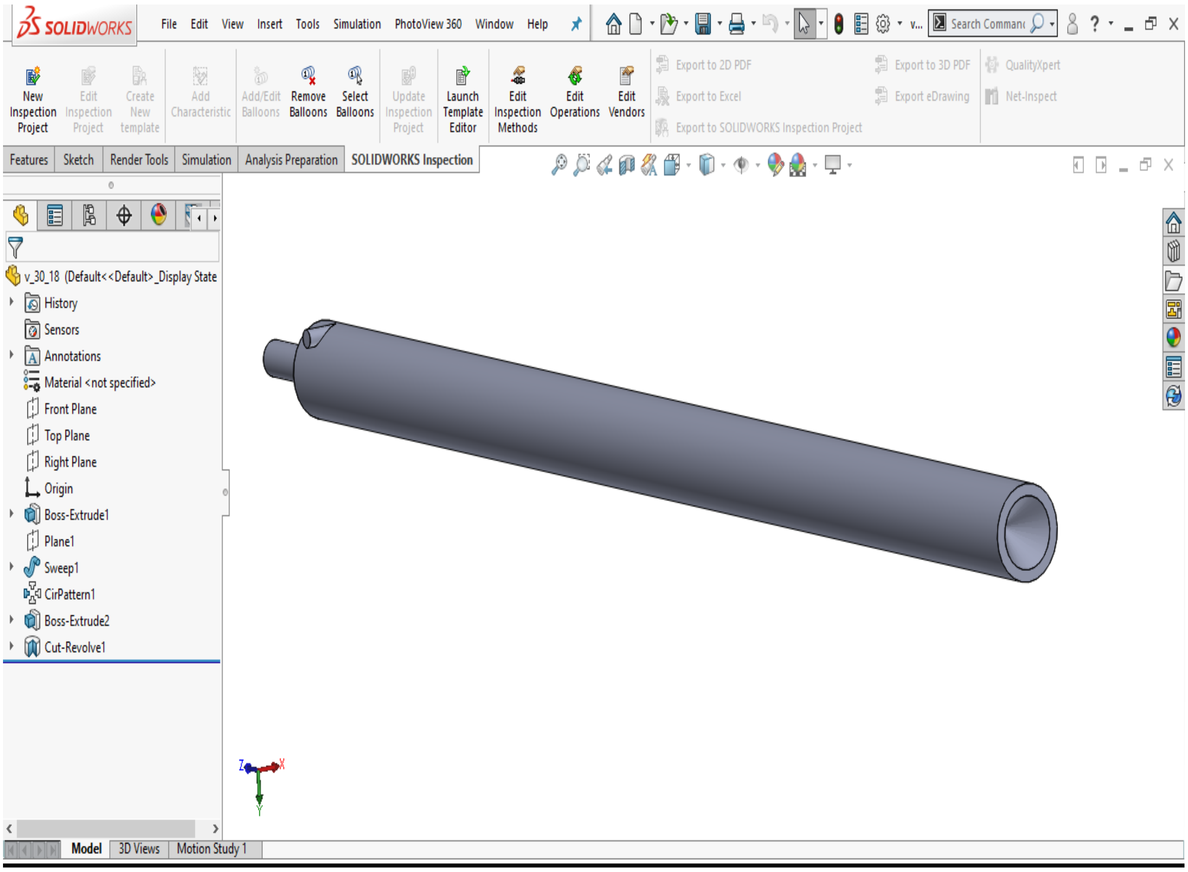


Figure 3.2: 3d model of vortex tube without insulation

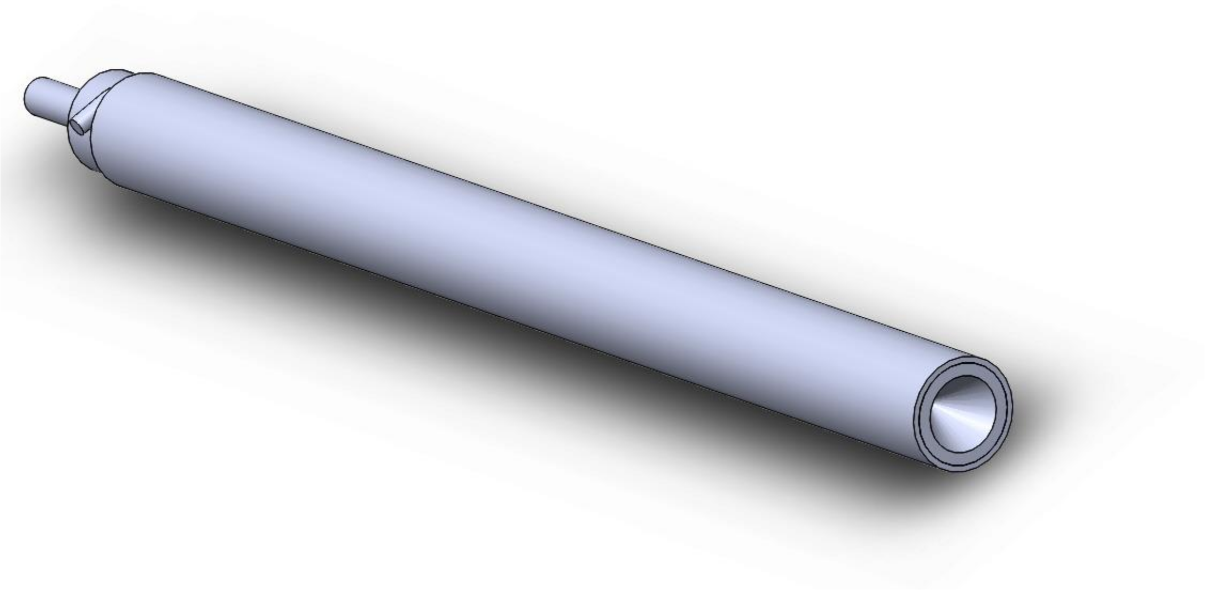


Figure 3.3: 3d model of vortex tube with insulation

3.15. Fluent(CFD)Analysis in Ansys

- “MESH RESULT WITH INSULATION
- TYPE ICEM CFD
  - Nodes 106087
  - Elements 22483

MESH RESULT WITHOUT INSULATION

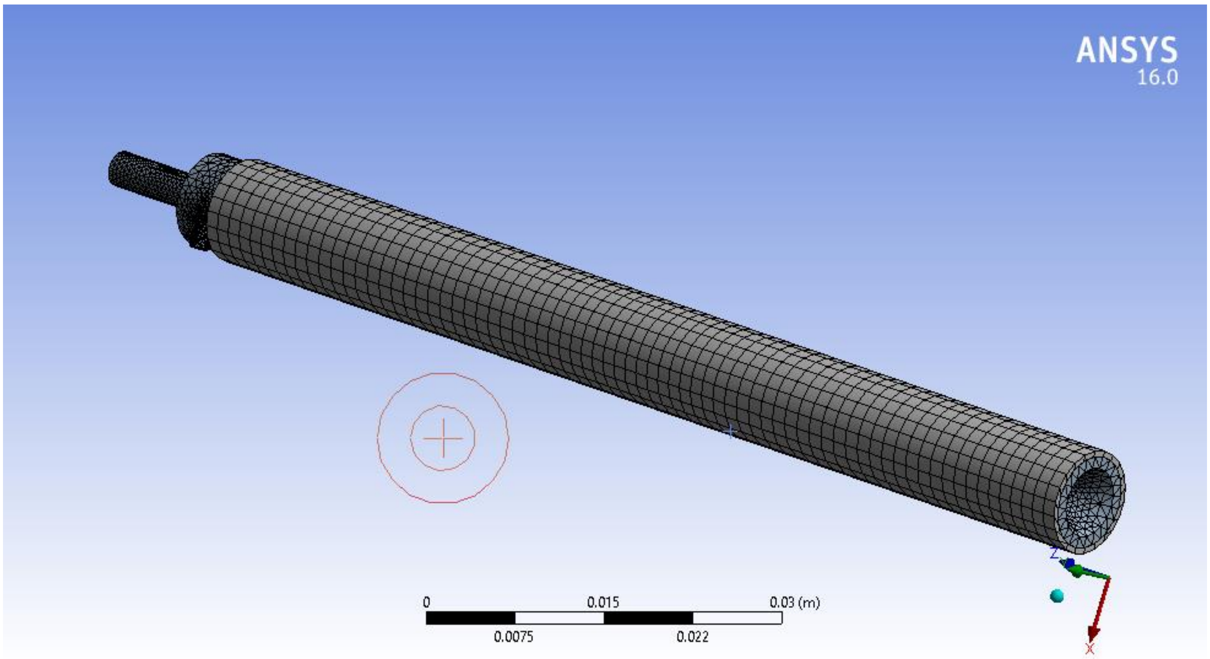


Figure 3.4: Meshing model of Vortex tube

Table3.2: Meshing details of model without insulation

	L/D=10	L/D=14	L/D=18
Nodes	15166	16577	18958
Elements	61525	65334	79163

- Nodes 106087
- Elements 22483

- ANALYSIS SETTING
- Analysis Type FLUENT
- Solver Target FLUENT

3.15.1. Solver settings: Assumptions:

- The working medium is ideal gas-air
- There is no heat interaction of the computation domain with the surroundings
- Flow is steady, turbulent and compressible
- Body force is negligible

3.15.2. Boundary conditions:

- Inlet :
- Pressure = 5 bar(gauge)
- Hot outlet :
- Pressure = 1 bar(gauge)
- Cold outlet :

CHAPTER 4

CALCULATION

4.1. Introduction

This section outlines the experimental method employed in the current literature to solve problems and gaps. Preliminary attention was paid to dimensional analysis and the construction of physical models and setup of the test plant. More details are then available on velocimetry techniques for the velocity monitoring of particles and velocity streaks, created according to whether an experiment is performed in the far or near field of the vortex.

Detailed test techniques are used to determine the effects of flow geometry of approaches, surface profiles, critical air core diameter, speed fields and flow visualisation. Finally, qualitative and quantitative observations are addressed on the expected relative insecurity of the test findings. We have done CFD ANALYSIS with different parameters

- At **tube length** or **L/D ratio- 10, 14,18** where tube diameter **D= 8mm,** so tube length will be respectively **80mm,112mm,144mm**.
- And at different **valve angle30^0,45^0,60^0**.
- And at different **inlet pressure 2 bar,3 bar,5 bar**.

  And using above parameters we got different **cold exit and hot exit temperatures** which are mentioned in the table.

4.2 InsulationonCopper VortexTube

Sincesurfaceroughnesscanaltertheperformance,insulationisprovidedontheoutsideof copper vortex tube while the internal surface is identical. The copper vortex tube isinsulated as shown in Fig.4.1.The thickness of insulation (35 mm) is selected such that the heat transfer from ambient to the periphery near inlet is reduced. Wax is chosen as an insulation material and the whole vortex tube is covered with the wax as shown in Fig. 4.1. This is achieved by casting wax over a copper vortex tube held horizontally.The vortex chamber is also casted with wax.By insulating the wholecopper vortex tube, it is expected that the performance of this insulated copper vortextube to be closer to the PVC vortex tube. Experiments are carried with air atP_o=2, 3, 5bar.

Figure4.1:Coppervortextubewithwaxcastingandcoveredbywooden material

4.3. CFD Simulation Analysis of Vortex tube in Ansys

In this section we studied the simulation CFD analysis of vortex tube at various parameters based on L/D ratio, various pressure Variations and insulation based and without insulation model analysis in Ansys.

Simulation figures shows the output variations in designed model, all figures are described below.

CFD Simulation analysis of Vortex tube at L/D: 10 at 2 bar pressure

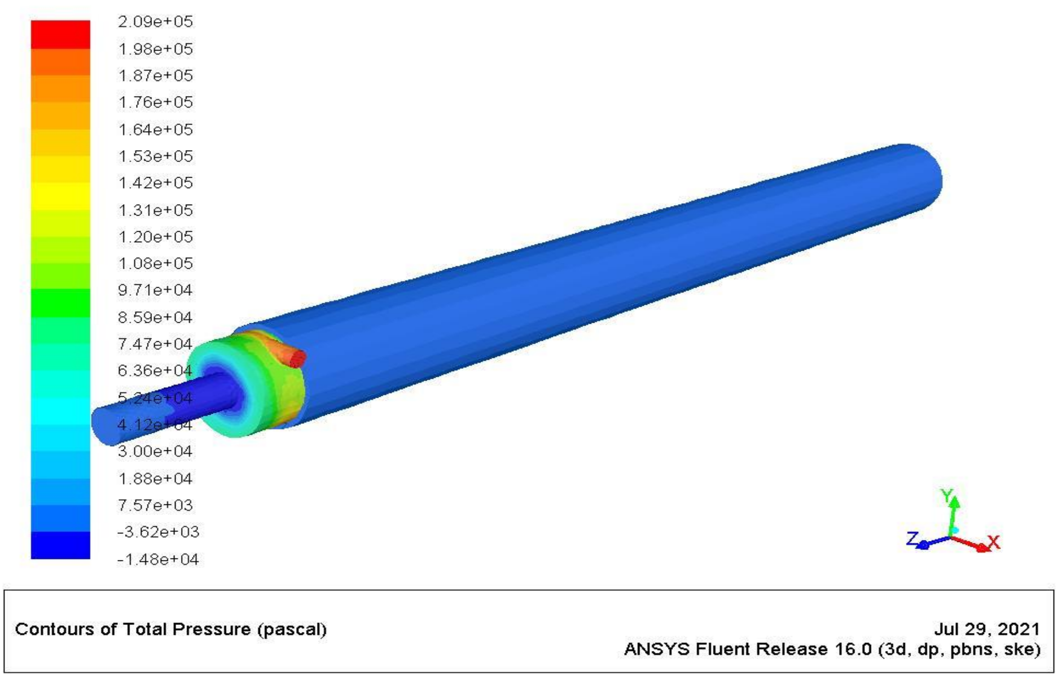


Figure 4.2: Total pressure of vortex tube model analysis at 2 bar pressures

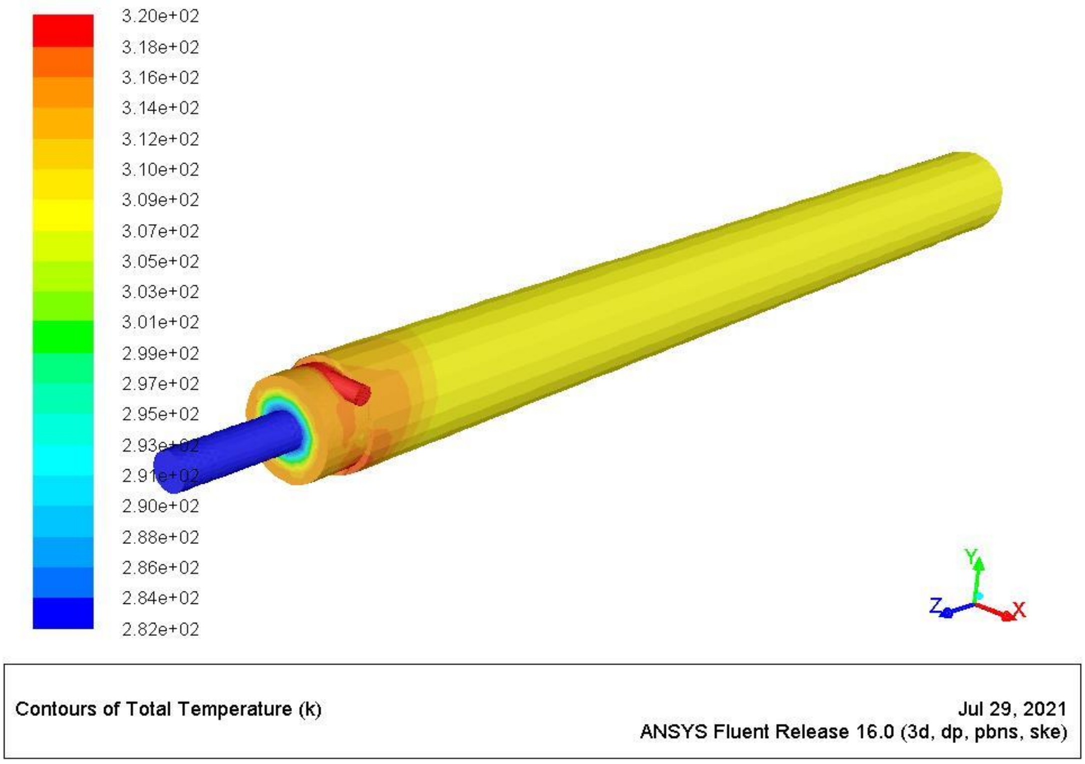


Figure 4.3: Total Temperature of vortex tube model analysis at 2 bar pressures

CFD Simulation analysis of Vortex tube at L/D: 10 at 3 bar pressure

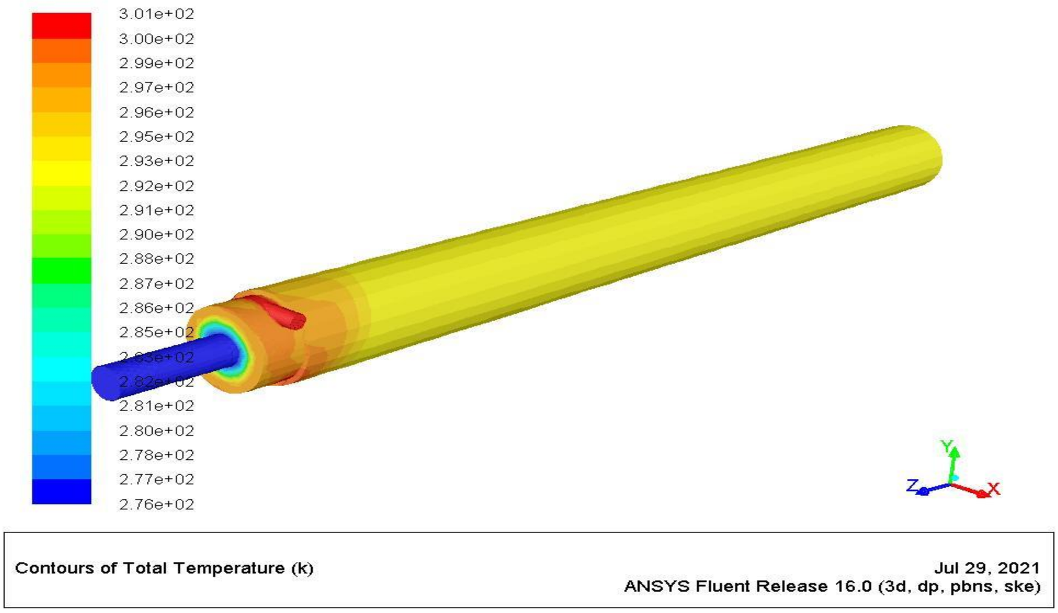


Figure 4.4: Total Temperature of vortex tube model analysis at3 bar pressure

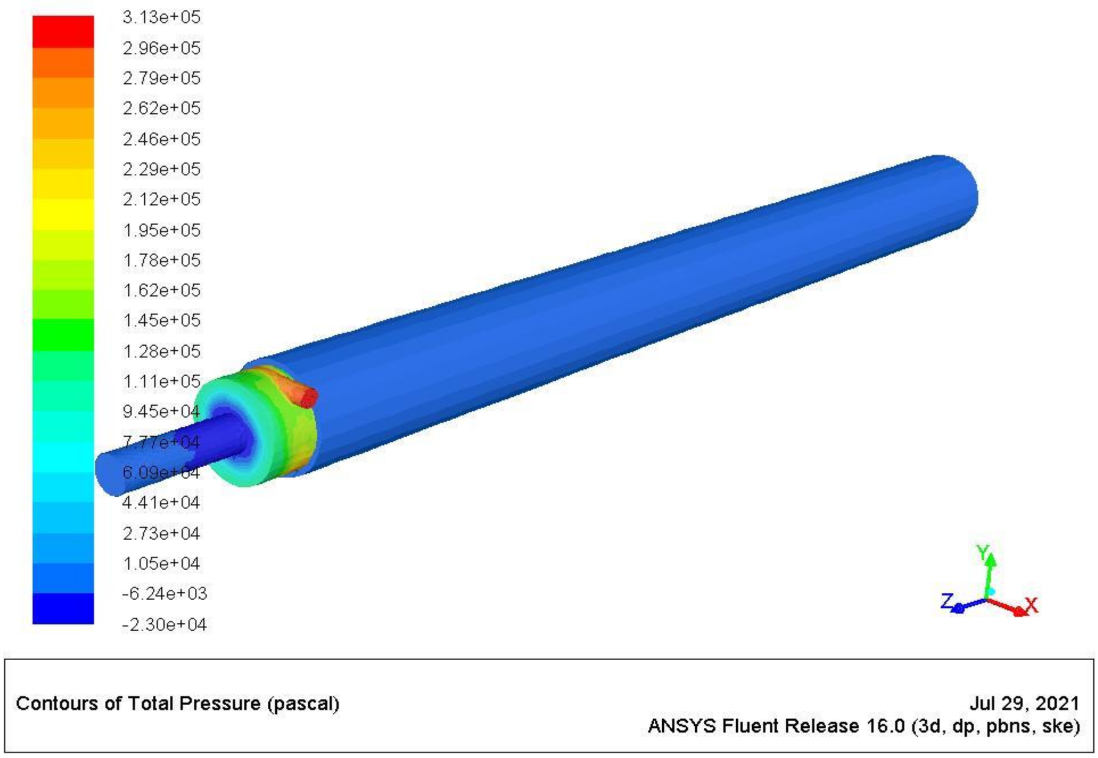


Figure 4.5: Total pressure of vortex tube model analysis at3 bar pressures

CFD Simulation analysis of Vortex tube at L/D: 10 at 5 bar pressure

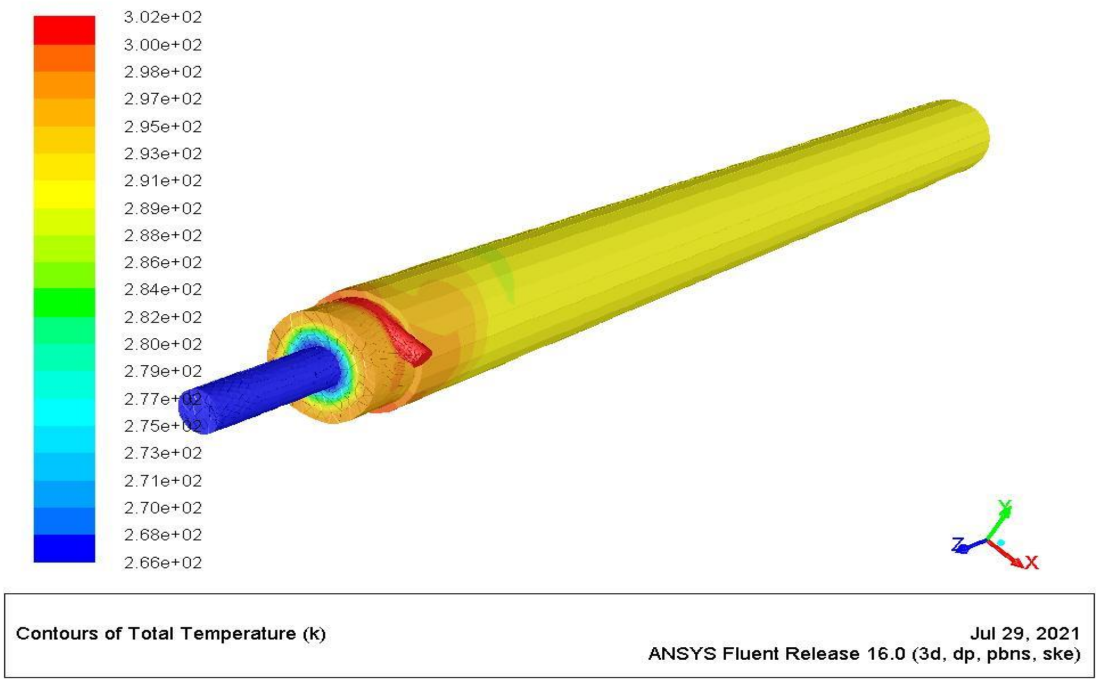


Figure 4.6: Total Temperature of vortex tube model analysis at5 bar pressure

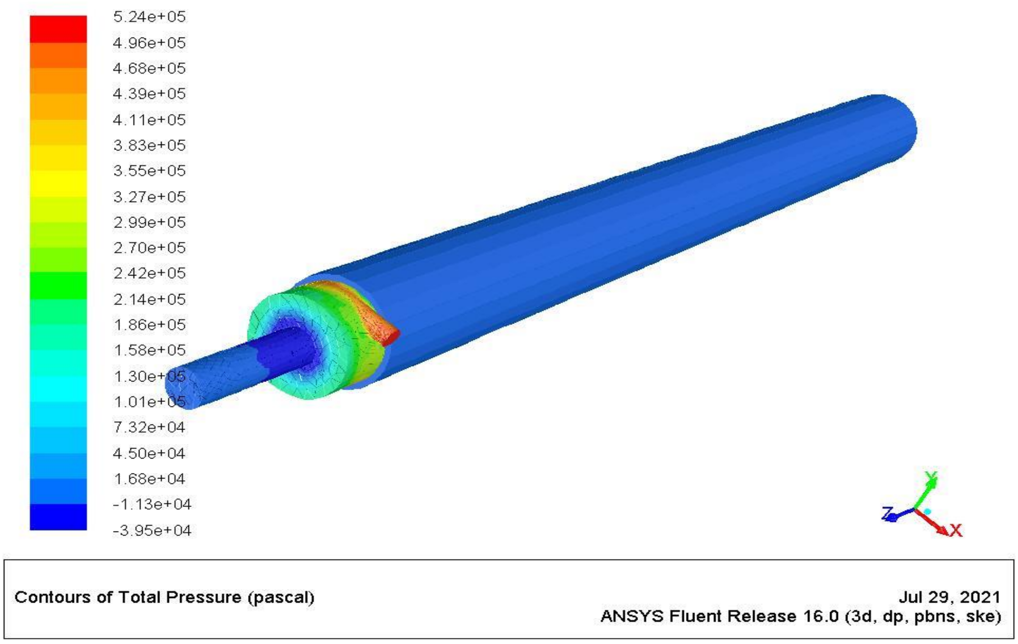


Figure 4.7: Total pressure of vortex tube model analysis at 5 bar pressures

4.4. Study of Output Temperatures (IN K)

We optimised the output data of temperatures from the vortex tube as a function of pressure changes of 2 bar, 3 bar, and 5 bar in this research.

Tables shows the temperatures in kelvin at various pressure are described below in tables.

4.5. TEMPERATURES (IN K)

In this section we compare the outlet temperatures of vortex tube at various L/D ratios.

Table 4.1: Without Insulation Temperature at Hot and Cold Exit

PRESSURE	VALVE ANGLE=30^0					
	L/D=10		L/D=14		L/D=18	
	COLD EXIT	HOT EXIT T	COLD EXIT	HOT EXIT	COLD EXIT	HOT EXIT
2 Bar	286	325	283	312	294	325
3 Bar	276	298	278	310	274	300
5 Bar	268	300	265	298	270	298

Table 4.2: With Wooden Material Insulation Temperature At Hot And Cold Exit

PRESSURE	**VALVE ANGLE=30^0**					
	L/D=10		**L/D=14**		**L/D=18**	
	COLD EXIT	**HOT EXIT T**	**COLD EXIT**	**HOT EXIT**	**COLD EXIT**	**HOT EXIT**
2 Bar	282	326	280	318	281	320
3 Bar	276	301	274	311	272	301
5 Bar	266	302	263	300	268	300

4.6. Pressure (IN PASCAL)

Study and analysed the effect of pressure variations in vortex tube at 2, 3, 5 bar of pressure and compare results in table 4.3 and 4.4 with various L/D ratios.

Table 4.3: Without Insulation Pressure At Exit

INLET PRESSURE	**VALVE ANGLE=30^0**								
	L/D=10			**L/D=14**			**L/D=18**		
	2 Bar	**3 Bar**	**5 Bar**	**2 Bar**	**3 Bar**	**5 Bar**	**2 Bar**	**3 Bar**	**5 Bar**
INLET PRESSURE	2.5e+4	3.57e+4	5.04e+4	2.76e+4	3.83e+4	5.17e+4	2.56e+4	3.46e+4	4.62e+4

Table 4.4: With Wooden Material Insulation Pressure At Exit

	VALVE ANGLE=30^0								
	L/D=10			**L/D=14**			**L/D=18**		
	2 Bar	**3 Bar**	**5 Bar**	**2 Bar**	**3 Bar**	**5 Bar**	**2 Bar**	**3 Bar**	**5 Bar**
INLET PRESSURE	3e+4	4.41e+4	7.32e+4	3.02e+4	4.65e+4	7.48e+4	3.18e+4	4.69+4	7.77e+4

4.7. Graphical Representation of Output parameters

Figures shows the graphical representation of variations in Temperature differences and pressure differences b/w insulated and non-insulated vortex tube model.

Graph for Non-insulated Vortex tube

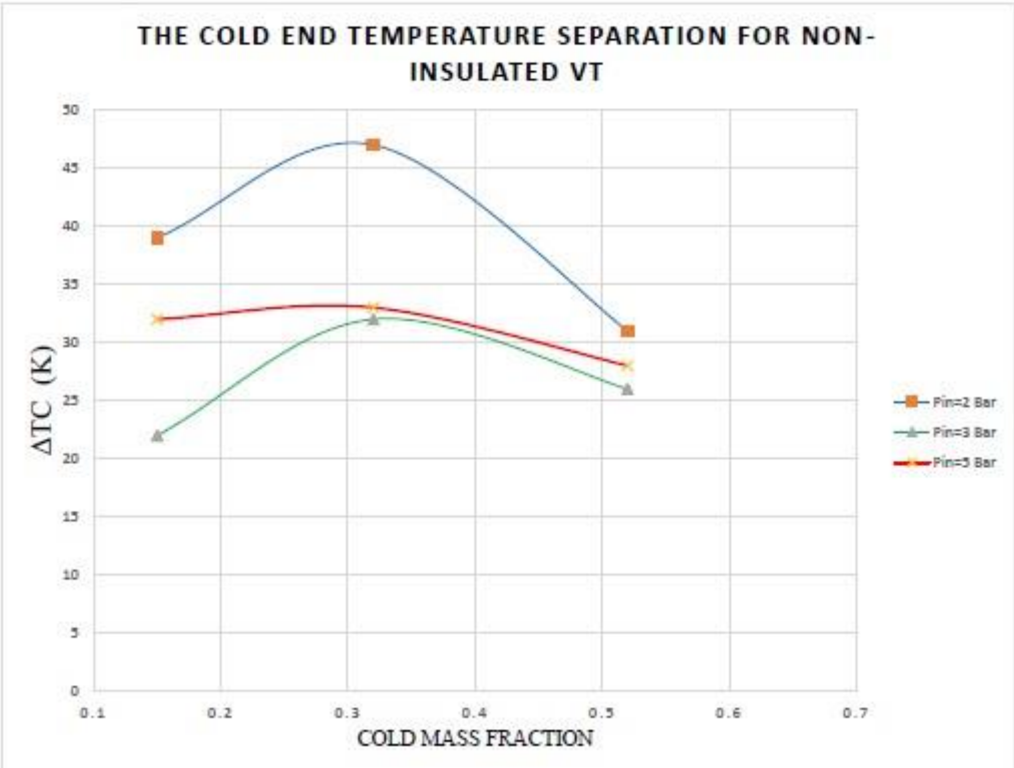


Figure 4.8: variations b/w temperature difference at various pressures in non-insulated

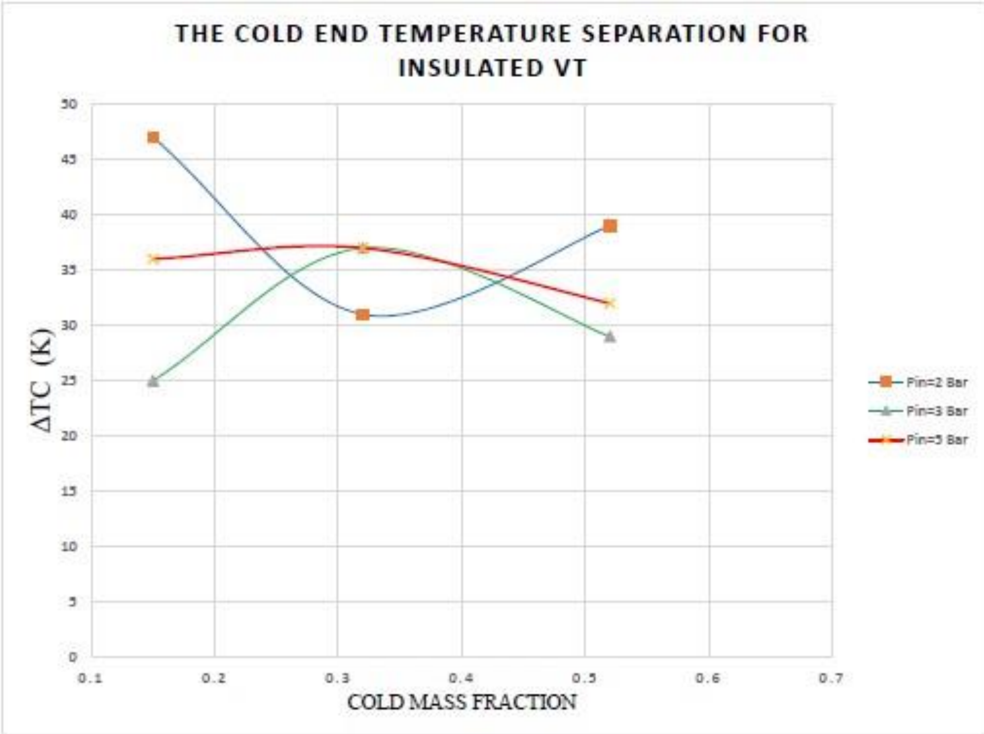


Figure 4.9: variations b/w temperature difference at various pressures in insulated

Comparison Between Non –Insulated and Insulated Vt At Different Inlet Pressure

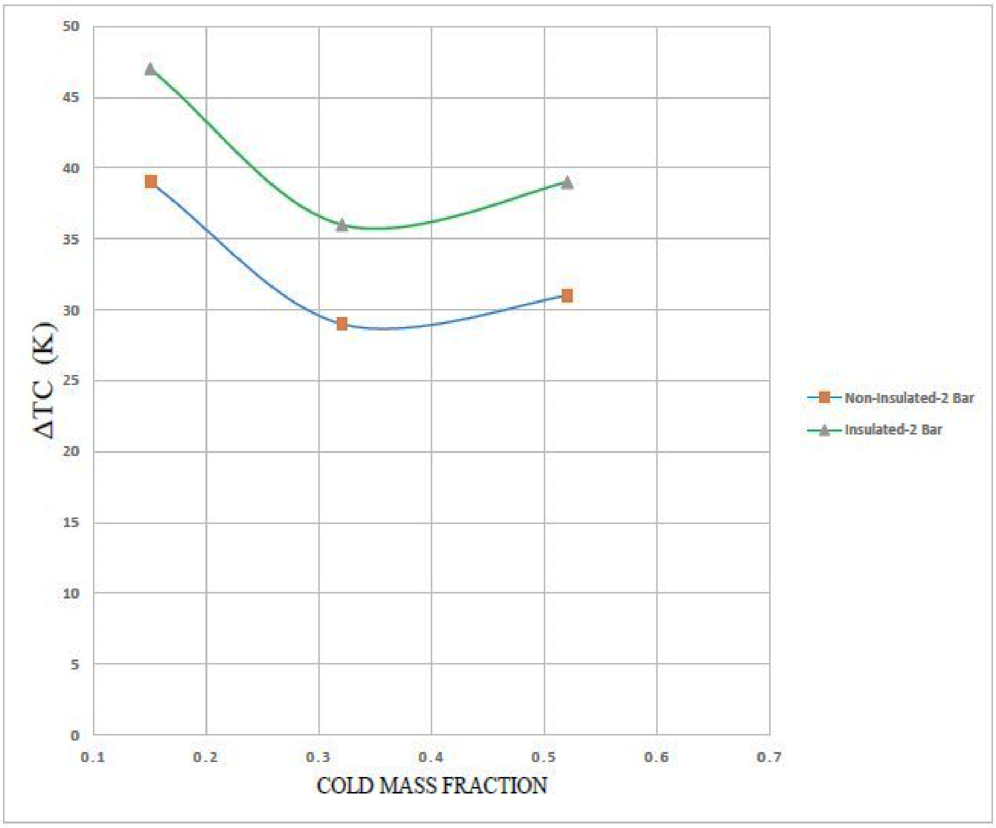


Figure 4.10: Temperature variation’s b/w insulated and non-insulated model at 2 bar

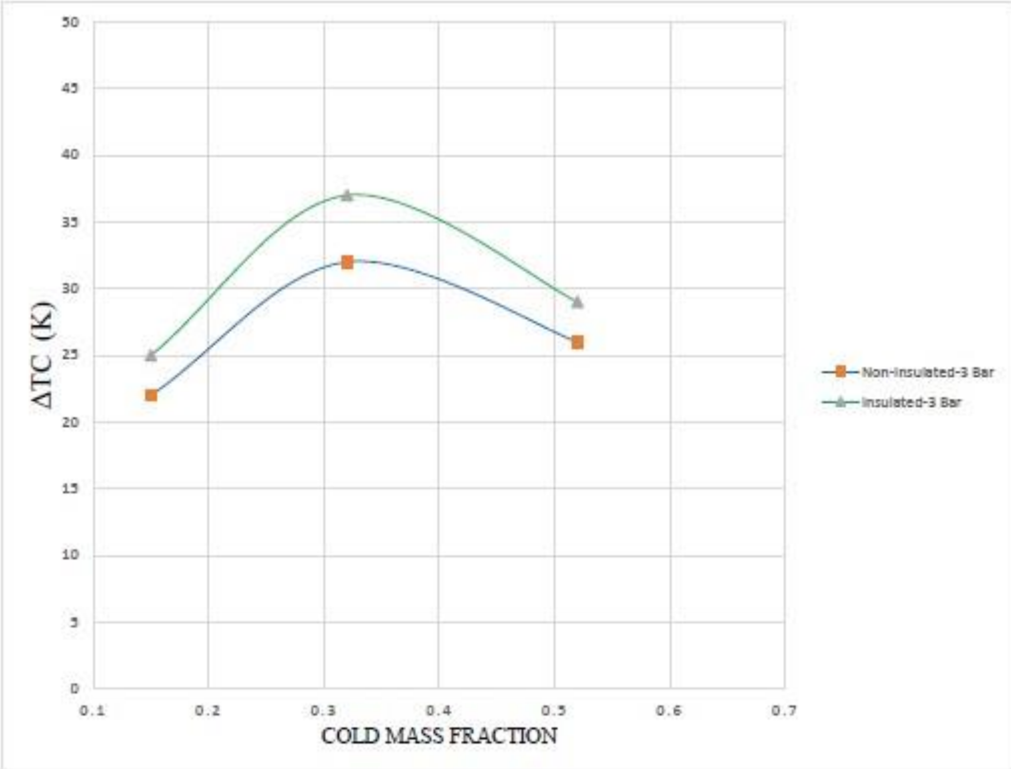


Figure 4.11: Temperature variation's b/w insulated and non-insulated model at 3 bar

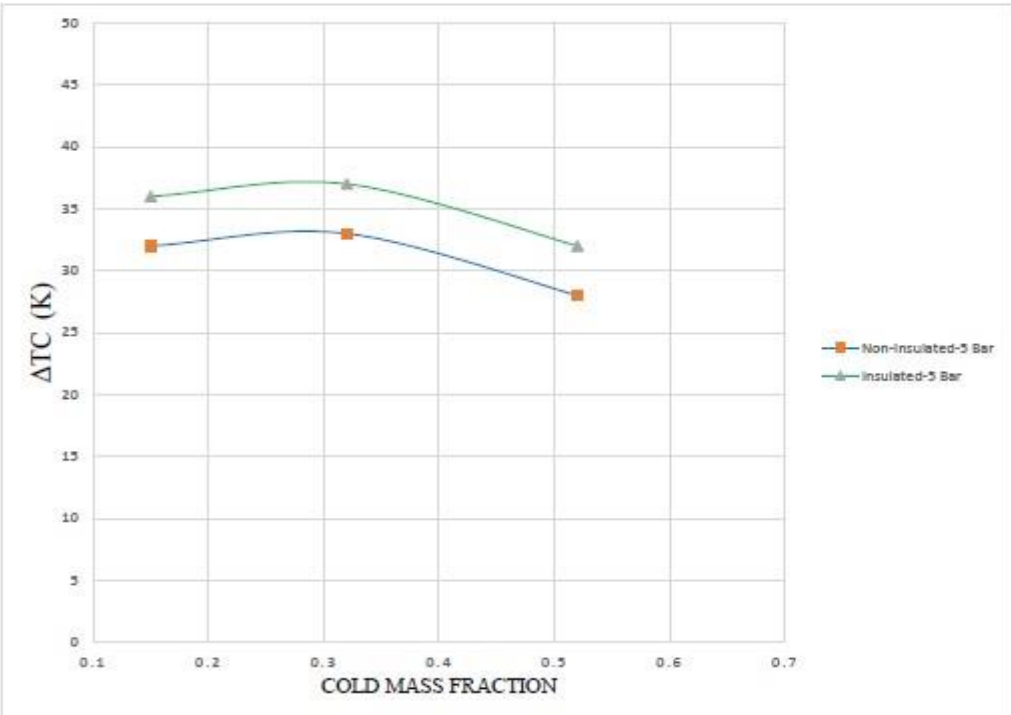


Figure 4.12: Temperature variation's b/w insulated and non-insulated model at 5bar

CHAPTER 5

CONCLUSION

5.1. Conclusion

In the current study, the CFD analysis of a vortex tube has been performed by using Ansys Tool with CFD analysis.We have done the modelling for the designed model by using Solid work software. We conducted CFD analysis at various input parameters of vortex tube fem analysis for vortex tube model in Ansys, While the mechanism of operation and energy separation in vortex tubes is still not defined, it is presently being developed for many industrial uses and is progressively becoming more diversified in the industry. In this numerical research, the energy separation process was investigated using a variety of critical factors affecting the performance of this device. It should be mentioned that for particular cooling applications, the vortex tubes examined in this research were evaluated. The findings of this research again demonstrated that the CFD is a helpful tool in the analysis of complicated streams and complex geometries to save time and money. Experimental findings from this research have shown that geometric form is very successful in its performance and have concluded that the cold outlet temperature separation, such as an umbilical cooling tube, is increased by the insulated and non-isolated divergency vortex tube.

- We have done CFD analysis at different inlet pressure 2 bar ,3 bar,5 bar.
- And we also did at different l/d ratio 10,14,18.
- And at both non-insulated & insulated vt condition.
- We got maximum temperature difference is 39 degree Celsius at 2 bar inlet pressure, L/d=10 in non-insulated vt.
- And in insulated vt the maximum temperature difference is 46 degree Celsius at 2 bar inlet pressure, L/d=10

REFERENCES

[1] Aghagolia, M. Sorin, "Thermodynamic Performance of a CO2 Vortex Tube based on 3D CFD flow analysis", International Journal of Refrigeration, Elsevier, 2019, doi: https://doi.org/10.1016/j.ijrefrig.2019.08.022.

[2] S. Deshmukh, K. D. Devade, "Exergy Analysis of Multi nozzle Vortex Tube", International Journal on Theoretical and Applied Research in Mechanical Engineering, V-7 Issue-1, 2018

[3] S. Deshmukh, K. D. Devade, "Exergy Analysis of Multi nozzle Vortex Tube", ResearchGate 2018.

[4] Ahmad Al-Qaisia, Jamil Al Asfar, Nbeel Abu Shaban, AreejEniezat, "Experimental Investigation of the Performance of a Vortex Tube with Conical Control Valve", Jordan Journal of Mechanical and Industrial Engineering, Volume 14, Number 2, June. 2020.

[5] D.G. Akhmetova, T.D. Akhmetov, "Flow structure and mechanism of heat transfer in a Ranque–Hilsch vortex tube", Experimental Thermal and Fluid Science, Elsevier, 113, 2020.

[6] Hong Yan ,Qingxiao Xu , Yongling Zhao, "The thermal performance of a novel convergent valveless vortex tube", International Journal of Refrigeration, Elsevier, 119, 2020.

[7] K. Dincer, Y. Yilmaz, A. Berber, S. Baskaya, "Experimental investigation of performance of hot cascade type RanqueeHilsch vortex tube and exergy analysis", Elsevier, international journal of refrigeration , 2011.

[8] K. Dincer, Y. Yilmaz, A. Berber, S. Baskaya, "Experimental investigation of performance of hot cascade type RanqueeHilsch vortex tube and exergy analysis", Elsevier, Internal Journal of Refrigeration, 34, 2011.

[9] Kiran D. Devade, Ashok T. Pise, "Exergy analysis of a counter flow Ranque–Hilsch vortex tube for different cold orifice diameters, L/D ratios and exit valve angles", Heat Mass Transfer, Springer, 2016.

[10] Kiran D. Devade, Ashok T. Pise, "Exergy analysis of a counter flow Ranque–Hilsch vortex tube for different cold orifice diameters, L/D ratios and exit valve angles", Heat Mass Transfer, Springer 2016.

[11] Konstantin, Matveev, Jacob Leachman,"Numerical investigation of vortex tubes with extended vortex chambers", School of Mechanical and Materials Engineering, Elsevier, 2019.

[12] Lizan Mahmood KhorsheedZangana, Ramzi Raphael Ibraheem Barwari, "The effect of convergent-divergent tube on the cooling capacity of vortex tube: An experimental and numerical study", Alexandria Engineering Journal, Elsevier, 59, 239–246, 2020.

[13] M. Yilmaz, M. Kaya, S. Karagoz, S. Erdogan, "A review on design criteria for vortex tubes", Heat and Mass Transfer, Springer, 2009.

[14] Maheswaran A, Purusothaman S, Shravanth Bharadwaj, "Design Of Vortex Tube And Analysis Of Its Flow Characteristics", International Journal of Engineering and Techniques -Volume 4, Issue 5, Sept - Oct 2018.

[15] Maheswaran, Purusothaman, Shravanth Bharadwaj, "Design Of Vortex Tube And Analysis Of Its Flow Characteristics", International Journal of Engineering and Techniques -Volume 4, Issue 5, Sept - Oct 2018.

[16] Muhammad FadhliSuhaimi, MohdHazwan Yusof, Mohamad Nor Hafiz Rashid, "The Effect of Tube Length, Cold Exit Diameter and Working Gas on The Cold Flow Temperature of Vortex Tube", MATEC Web of Conferences 225, 02010, 2018. available at: https://doi.org/10.1051/matecconf/201822502010

[17] R Liew, J C H Zeegers, J G M Kuerten, "Temperature, Pressure and Velocity measurements on the Ranque-Hilsch Vortex Tube", 6th European Thermal Sciences Conference, Journal of Physics: Conference Series 395, 2012, 012066.

[18] S.Karthik, "Design and Computation of COP of Vortex Tube", International Journal of Scientific & Engineering Research, Volume 6, Issue 4, April-2015.

[19] Seyed Ehsan Rafiee, Masoud Rahimi, "Experimental study and three-dimensional (3D) computational fluid dynamics (CFD) analysis on the effect of the convergence ratio, pressure inlet and number of nozzle intake on vortex tube performanceeValidation and CFD optimization", Energy, Elsevier, 2013.

[20] Seyed Ehsan Rafiee, Masoud Rahimi, "Experimental study and three-dimensional (3D) computational fluid dynamics (CFD) analysis on the effect of the convergence ratio, pressure inlet and number of nozzle intake on vortex tube performance Validation and CFD optimization", Energy, Elsevier, 2013.

[21] T. Dutta , K.P. Sinhamahapatra , S.S. Bandyopadhyay, "Experimental and numerical investigation of energy separation in counterflow and uniflow vortex tubes", International Journal of Refrigeration, Elsevier, 2020.

[22] T. Duttaa, K. P. Sinhamahapatrab, S. S. Bandyopadhyay, "Experimental and numerical investigation of energy separation in counterflow and uniflow vortex tubes", International Journal of Refrigeration, Elsevier, 2020.

[23] Upendra Behera, P.J. Paul, S. Kasthurirengan et, al.,"CFD analysis and experimental investigations towards optimizing the parameters of Ranque–Hilsch vortex tube", International Journal of Heat and Mass Transfer, Elsevier, 2005.

[24] K. Kiran Kumar Rao, Dr. G. Sharanappa and Dr. A. Ramesh, "EXPERIMENTAL ANALYSIS OF VORTEX TUBE BY USING DIFFERENT MATERIALS", International Journal of Mechanical Engineering and Technology (IJMET) Volume 9, Issue 9, September 2018, pp. 1173–1181, Article ID: IJMET_09_09_128 Available online at http://www.iaeme.com/ijmet/issues.asp?JType=IJMET&VType=9&IType=9 ISSN Print: 0976-6340 and ISSN Online: 0976-6359 © IAEME Publication Scopus Indexed.

[25] K.K.Arun and S.Tamilselvan, "Optimizing the Parameter of The Vortex Tube Through CFD Analysis For Sustainable Manufacturing", IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE) e-ISSN: 2278-1684, p-ISSN: 2320-334X PP 06-12 www.iosrjournals.org.

[26] Khait, A., Noskov, A., Alekhin, V., &Antipin, A. (2013). Numerical Simulation and Visualization of Air Flow in Ranque-Hilsch Vortex Tube 1. 13th International Conference on Construction Applications of Virtual Reality, 30-31 October 2013, London, UK, October, 30–31.

[27] Krishna Kumar Karothiya and Siddharth Chauhan, "FABRICATION AND ANALYSIS OF VORTEX TUBE REFRIGERATION SYSTEM", International journal of innovative research in science and engineering vol. no2, issue 08 august 2016.

[28] Madhu Kumar, R., & Sudheer, N. V. V. S. (2019). Computational fluid dynamics study on the effects of L/D ratio in a 2 stage hot cascade vortex tube. International Journal of Recent Technology and Engineering, 7(ICETESM18), 121–124.

[29] Manickam, M., &Prabakaran, J. (2019). CFD analysis on Ranque–Hilsch vortex tube with different cold orifice diameter and cold mass fraction. International Journal of Ambient Energy, 0750. https://doi.org/10.1080/01430750.2019.1681292.

[30] Maheswaran A, Purusothaman S and Shravanth Bharadwaj C, "Design Of Vortex Tube And Analysis Of Its Flow Characteristics", International Journal of Engineering

and Techniques -Volume 4, Issue 5, Sept - Oct 2018 ISSN: 2395- ISSN: 2395-1303 1303 1303 http://www.ijetjournal.org.

[31] Matveev, K. I., &Leachman, J. (2019). Numerical investigation of vortex tubes with extended vortex chambers. International Journal of Refrigeration, 108, 145–153. https://doi.org/10.1016/j.ijrefrig.2019.08.030.

[32] Mohammad O. Hamdan, Ahmed Alawar, Emad Elnajjar and Waseem Siddique, “Experimental analysis on vortex tube energy separation performance”, Received: 18 September 2010 / Accepted: 25 May 2011 / Published online: 8 June 2011 Springer-Verlag 2011 Heat Mass Transfer (2011) 47:1637–1642 DOI 10.1007/s00231-011-0824-6.

[33] Mukesh Kumar, P. C., & Chandrasekar, M. (2019). CFD analysis on heat and flow characteristics of double helically coiled tube heat exchanger handling MWCNT/water nanofluids. Heliyon, 5(7), e02030. https://doi.org/10.1016/j.heliyon.2019.e02030.