Exergy and energy analysis of vortex Tube

A Thesis Submitted

In partial fulfillment for the reward of the degree of

Master of technology

In Mechanical Engineering

SUBMITTED BY

Nikunj kumar Goyal

(2K19/THE/15)

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, NIKUNJ KUMAR GOYAL, here by certify that the work which is being presented in thesis entitled "Exergy and energy analysis of vortex Tube" 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.

NIKUNJ
30/08/2021
NIKUNJ KUMAR GOYAL

(2K19/THE/15)

CERTIFICATE

I, NIKUNJ KUMAR GOYAL, hereby rectify that the work which is being presented in this thesis entitled “Exergy and energy analysis of vortex Tube” in the partial fulfillment 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 endeavors.

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 moral support and motivation to complete this project in due course.

NIKUNJ
30/08/2021
Nikunj Kumar Goyal

(2K19/THE/15)

ABSTRACT

In this study performance of Vortex tube coupled with Vapour compression refrigeration system analyzed using different vortex tube length, valve angle and L/D ratio by using exergy and energy analysis. It is a mechanical device with no moving components. The splitting of the flow into low- and high-temperature zones is called the temperature separation effect. The output of the vortex tube is determined by two main parameters: one is the working parameter, for example, intake pressure for compressed air; the other is geometric parameters, such as nozzle number, nozzle diameter, cone valve angle, hot side tube length, hot opening diameter and vortex tube material. The Vortex tube has fascinating functionalities and many industrial applications and is employed as a cooling unit in industry as a refrigerator.

"The purpose of this study is to illustrate the geometric factors that influence vortex tube performance, and the analytical effect of CFD. This paper includes equations and boundary criteria for examining the vortex tube. Hypertension, viscosity, turbulence, temperature gradient, and secondary circulation are all examples of temperature separation. Furthermore, studies have shown that CFD analysis assesses various kinds of nozzle profiles as well as the number of nozzles. CFD analysis was performed to establish the highest warm-gas temperature and the lowest cold-gas temperature using different shapes and dimensions of the hot end valve. The boundary conditions of the necessary vortex flow have been modified."

TABLE OF CONTENTS

LIST OF TABLES

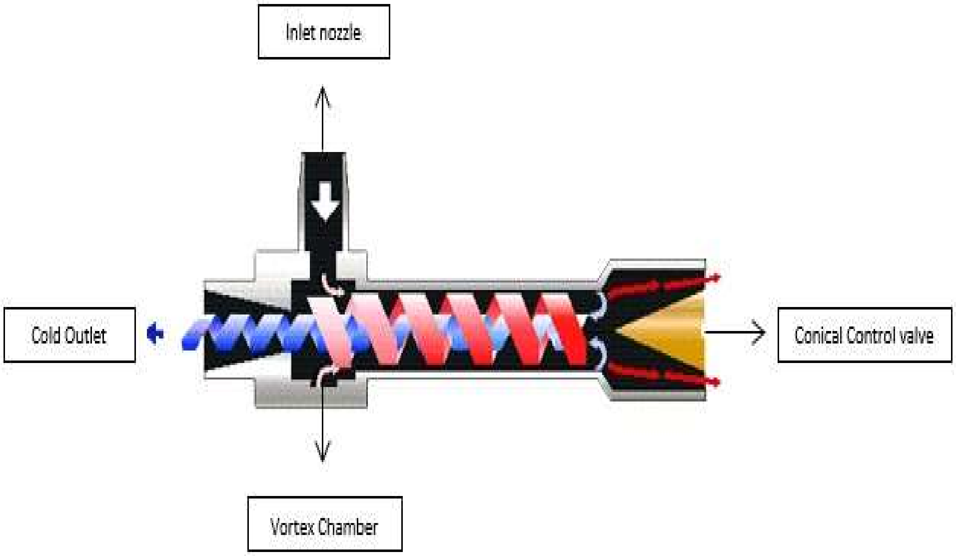
LIST OF FIGURES

CHAPTER 1

INTRODUCTION

1.1. Introduction

In 1932 Ranque invented the Vortex tube. It acts as a device able to separate compressed air supplied by various mass fractions into cold and hot streams. This process is known as the separation of energy (temperature). An axial tube having a vortex chamber, a cold gas outlet and the hot gas outlet comprises one or more of the tangential entrance nozzles. Air without dehumidification from dust between 5 and 6 Bars is tangentially fed via the intake nozzles into the vortex chamber. The air expands via the jacks and reaches a high angular velocity, resulting in a high swirling flow, and with the conical control valve, the gas travels towards the heating outlet (shown in figure 1a). A part of the gas flowing in the heating outlet reverses its course and proceeds to the cold outlet. It travels in the axial direction of the tube to the cold outlet.



(a) General model of Vortex tube

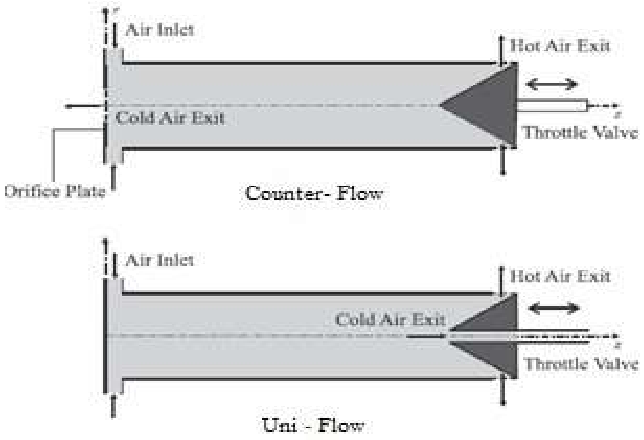


Figure 1.1: The Vortex tube: Schematic diagram [2]

“As the flow travels to the cold outlet, the heat energy in the flow reversal is transferred to the flow's perimeter. The peripheral flow becomes warmer as the input gas and the core flow area become colder than the inlet gas. The peripheral flow is blocked by the ring gap between the tube wall and the conical valve at the hot outlet. The cold exit is where the core flow exits. The relative mass flow rate of hot and cold gases is controlled by a conical valve on the hot outlet. The Ranque effect is the name for these energy separation phenomena. The vortex tube can produce a temperature of 127 °C with a heat flow and a cold air flow of -46 °C with a heat flow.

This design is termed the counter-flow design of the vortex tube with cold aperture and conical valve on the opposing sides. Figure 1b illustrates a different kind of vortex tube. It is termed the vortex tube's uni-flow version. In this arrangement just one end of the vortex tube leaves the cold and warm air streams. The hot valve and the cold aperture are on the same side of the vortex tube in this kind of construction. The other one is completely closed on the vortex tube. The Uni-Flow design vortex tube is less efficient than the anti-flow design vortex tube since streams coming out of the exhaust are mixed. Thus, the vortex tubes are chosen for commercial applications in the counter flow design”.

The RHVT is a genius attributed by Georges Joseph Ranque and Rudolf Hilsch, who separately designed the weapon during the war, which in the 1940s devastated European countries [1].

An injected gas stream is divided into two streams by RHVT which is considerably hotter, while the inject able gas stream is significantly cooler. This is noteworthy since moving components or labour elements are not present. The fluid dynamic and thermodynamic processes of a RHVT are highly complicated, but geometrically basic. It is extremely difficult to test for a greater insight into the heat migration mechanism and monitor velocity, pressure and temperature contours precisely inside the tube. The use of CFD is a simpler option. There is yet no convincing proof of the energy migration mode inside the RHVT.

1.3. The Ranque - Hilsch Vortex Tube

British physicist James Maxwell proposed during the 19th century that if one little daemon could be captured and trained to open and shut a small valve a system might be designed to extract warm and cold water from a single pipe. Only if a rapid (hot) molecule approached it, the demons would open the valve and shut the valve against slow (cold) molecules[2]. This hypothetical gadget may provide a source of warm and cold fluids on demand at ambient temperatures with assistance from Maxwell's devil. Although Maxwell hypothesised, he may be shocked to discover that such an apparatus will become a reality in the following century. This gadget, whose first name was Maxwell's Demon Tube, eventually became known today as the RHVT. The cross part scheme illustrated in fig. 1-1 shows very clearly that, despite the lack of moving parts or work inputs, the RHVT is really an apparatus that concurrently separates the gas flow into two streams, considerably hotter and cooler than intake temperature. The main reason for this absence of a traditional energy source is in theory why scholars and fans are so keen in the gadget.

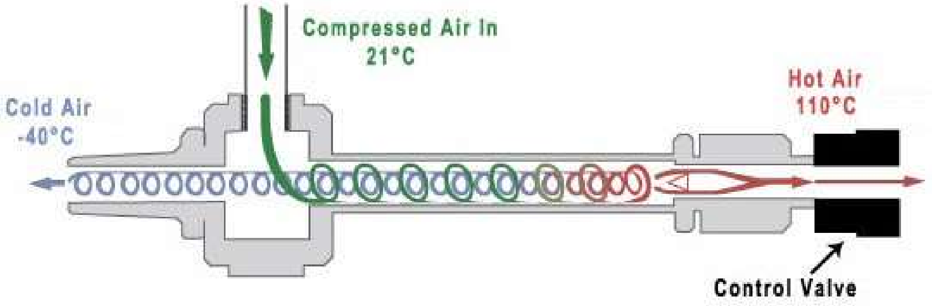


Figure. 1.2: Typical Operation of a Modern Counter-Flow RHVT [3].

As mentioned above, the genesis of RHVT was acknowledged by the Frenchman Georges Joseph Ranque and the German Rudolf Hilsch, during and after the period of the war in Europe in 1933-1946. They will now explain their contributions.

1.4. Ranque's Contribution

"Soon after, C.D. Fulton[1] published one of the most comprehensive historical articles detailing the chronology of events leading up to and analysis of the RHVT's discovery. When Ranque, the technique's discoverer, appeared before the French Physicist Society in June 1933, he told the general audience that hot and cold air from opposite ends of the pipe were greeted with mistrust. At the time, aerodynamicists simply concluded that stagnation temperature and static temperature were interchangeable." [1].

The image below was taken from the patent of Ranque, along with others in this section. [4].

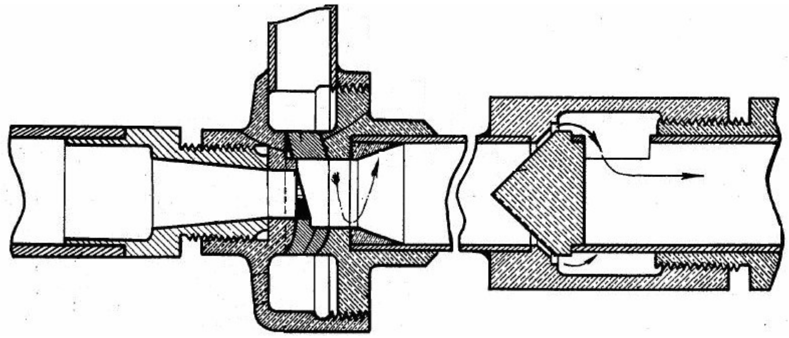


Figure. 1.3: Cross-Section of Ranque's Vortex Tube Design [4].

Commercial vortex tubes are basically quite similar to those depicted in Fig. 1-2, which is demonstrated by the explosion of the contemporary RHVT in Fig. 1-3, below.

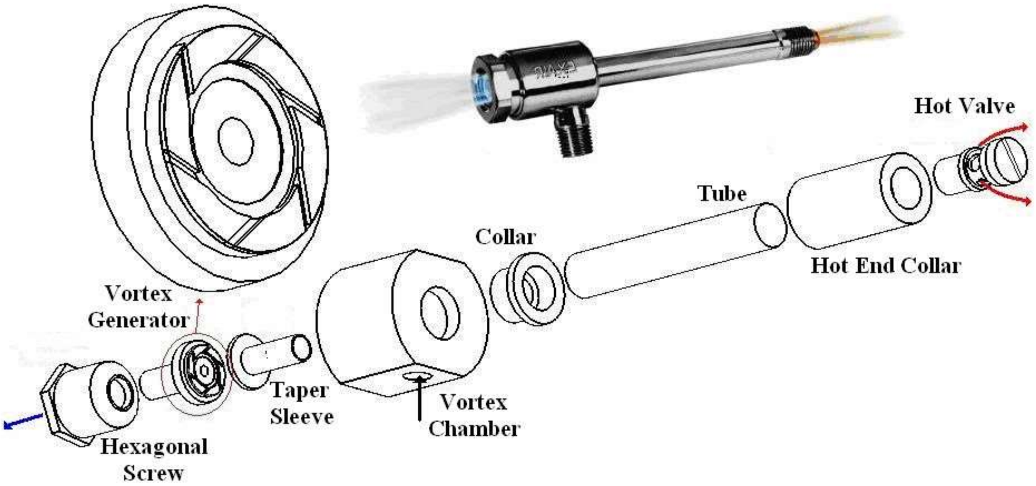


Figure 1.4: Commercial RHVT by Exair [5].

“On 12 December 1931 when Ranque submitted a French patent docket, the first public document concerning the RHVT phenomena was published. He filed the identical dockets in the United States after a French patent was granted in 1931 as stated before. In this invention the Ranque demonstrated that a single nozzle, a variety of nozzles or a set of blades may be the tangential entrance. He also described how, by adjusting the size of the cold-air or limiting the end of the hot tube you can achieve a small amount of moderately cold air and said that when the end of the hot tube is completely closed, the colder the cold air will be, the higher the pressure of the air supplies, the higher the cold air. Ranque also spoke about measuring the pressure distribution, a job that is much simpler to say than to perform, as seen in Fig. 2-1.

The theory Ranque gives in the United States patent, was stated as follows”, “The spinning gas stretches over the wall of the tube in a thick sheet and presses centrifugal force in the inner layers of this sheet, heating them. The inner layers expand and get chilly at the same time.

Friction needs must be minimized between the layers” [4].

1.5. Hilsch’s Contribution

Hilsch, a physicist at the University of Erlangen, is said to have learned of this device through information provided to him from occupied France[1]. In 1946, Hilsch published an article[6] in which he briefly mentioned the Ranque's 1933 study as the source of the idea. Despite this, he arrived at the same design as Ranque's drawings. [1].

In this article Hilsch wrote: “The air passing through the orifice has been expanded in the centrifugal field from the region of high pressure at the wall of the tube to a low pressure near the axis. During this expansion it gives considerable part of its kinetic energy to the peripheral layers through increased friction. The peripheral layers then flow away with increased temperature….The internal friction….causes a flow of energy from the axis to the circumference by trying to establish a uniform angular velocity across the entire cross section of the tube” [6].

“Following Hilsch's lead, virtually everyone has used the same counter flow designs, such as those shown in Figure 1-4 below, regardless of uniflow type. This is partially due to the fact that a counter flow type is easier to manufacture and offers two distinct warm and cold outputs at opposite ends of the tube”, preventing one thermal output from interfering with the other. As a consequence, this project is primarily concerned with counter-flow design.

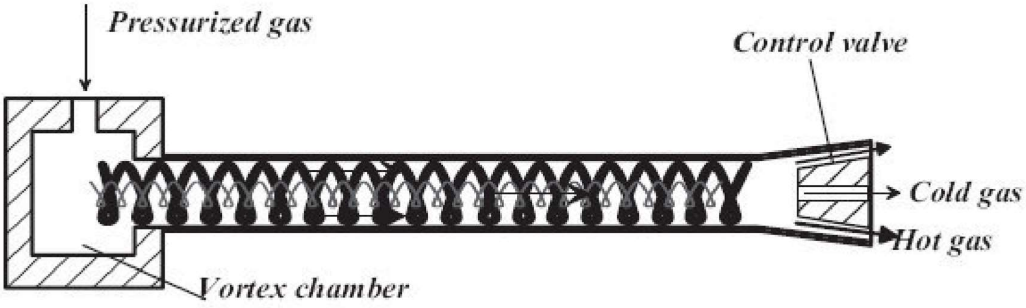


Figure. 1.5: Cross Section of a Uniflow RHVT [7].

1.6. Energy Separation Phenomenon In Vortex Tube

"Despite the fact that several scholars have focused their attention on understanding energy separation in the vortex tube, there have been no competition for the theoretical explanation of the energy separation phenomena or the vortex effect. Some opinions on these phenomena are addressed below in previous works.

As one air stream travels up and down, the two spin at the same angular speed in the same direction. This means a particle in the inner stream completes one revolution of a particle in the outer stream in the same period of time. The rotational speed of the smaller vortex may nevertheless be anticipated to rise according to the concept to preserve the angular momentum". But the velocity of the inner vortex stays same in the Vortex tube. Angular dynamic from the inner vortex was lost. The wasted energy in the external vortex becomes heat. This warms the outside vortex and cools the inside vortex.

A second perspective shows that the tube is comparable to the "solid body rotation," meaning that the inner vortex's rotational rate (angular velocity) is the same as the external vortex. This perspective disagrees with the conventional vortex behavior of many previous studies, in which internal fluid spins at a greater rate than external Fluid. Probably due to the lengthy period each part of the air remained in the turret, the solid body rotation has a significant impact on the friction between the inner parcels and the outside parcels. It is generally accepted that a little trend towards the centre is hot air, although the impact is insignificant if the turbulence is diminished.

An additional simple reason is the greater pressure of the external vortex than the inside vortex (because of centrifugal force). The temperature of the external vortex is thus greater than the inside vortex.

1.6.1 Internal Flow Field

The flow field research inside the Vortex tube began with the use of flow visualizing methods like liquid and smoke.

Inside the Vortex tube Lay (1959) was injected with water and nothing was disclosed. A combination of powdered carbon and oil was employed by Sibulkin (1962). The Smoke utilized as an input medium by Smith (1962). The mass-ratio of kerosene utilized in Piralishvili and Polyaev (1996) was 1:30. All the studies focused on following the flow route on the end-wall by means of a Lucite tube as a vortex tube to follow the liquid components with liquid injection. The velocity distribution from the flow path on the end wall is like strong body rotation in the centre. Smoke visualization illustrates the flow route inside the axis tube. The flux pattern within the pipe may be differentiated from the two distinct portions of the swirling flux: the periphery part of the area with an axial movement to the hot end, and the centre part with axial movement toward the cold end. The flow field within the tube is extremely easy to measure qualitatively using such imaging methods. However, quantitative flow information and the temperature field inside the tube are not promising. The methods of visualization only assist the quality description of the flux field within the Vortex Tube. Appropriate samples are required for further information on flow in relation to pressures, temperatures and velocity within the system. For temperature measurements by Smith and Reynolds in 1962 and Gao et al in 2005, a Pitot tube was used in velocity and thermocouple.

The aim of these experimental studies is, in conclusion, to discover empirical terms that may be utilized for optimizing the RHVT system. The purpose is to use the RHVT for broader applications, such as cooling and heating of room and components, cleaning, purifying and gas separation.

1.6.2 Controlling Temperature and Flow in a Vortex Tube

“Cold air flow and temperature may be readily controlled using a controlled conical valve in the hot air outlet. By opening the valve, the cold flow and cold air temperature are reduced. The valve closure, on the other hand, increases the flow of cold air and the temperature of cold air". The "cold fraction" refers to the air percentage at the Vortex Tube's chilly exit. "In most applications, a cold fraction of approximately 80% produces a combination of cold flow and temperature reduction that optimizes cooling. The cold air flow rate is impaired, even if the lowest temperatures with low cold (less than 50%) fractions are produced. Process cooling, component cooling and chamber cooling are all sectors that need maximum cooling. Simply place a thermometer in the cold air exhaust and adjust the conical valve on the hot end to set the Vortex Tube for its variable capacity (cooling and heating)".

1.6.3 Temperature Separation Effect

"There are two types of vortex tubes: free and forced. In a free vortex, a fluid particle's angular speed increases as it moves toward the centre of the vortex (like a whirlpool). The closer a fluid particle is to the vortex's centre, the faster it spins. In a forced vortex, the velocity is proportional to the radius of the vortex. The particle speeds slow down when the fluid particle gets closer to the vortex centre. In a Vortex tube, the movement of the outer (hot) air stream is a free vortex. Inner (cold) air moves in a forced vortex. The rotating movement of the forced vortex is controlled by the free vortex (hot air stream). Both cold and warm air streams turbulence results in a single spinning stratum mass. Through the hollow core of the exterior air stream, the internal air stream moves at a slower rate than the external air stream. Because energy is proportional to the quadrature of speed, the cold air stream loses energy via heat transfer. As a result, energy may be transferred from the inner air to the outside air stream as heat, resulting in a chilly inner air stream".

1.7. Main Parts of a Counter-Flow Vortex Tube

The main parts of the counter-flow Vortex Tube used in the present work are shown in Figure 1.2.

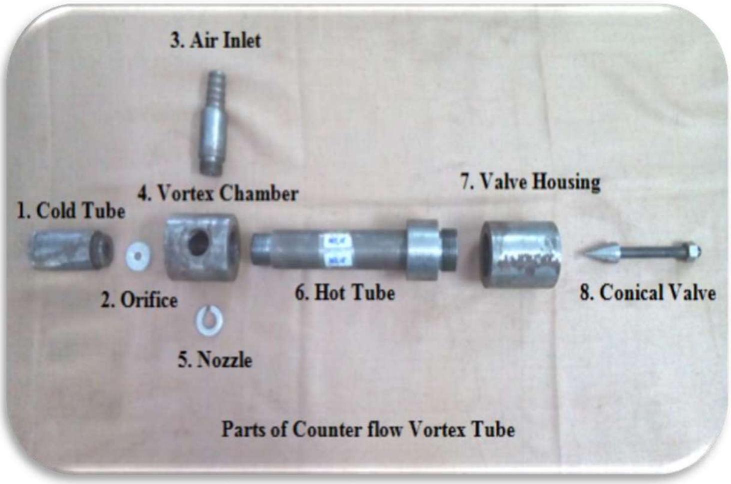


Figure 1.6 Main parts of counter-flow Vortex Tube

A Vortex tube using compressed air as a power source does not have moving components and generates cold air from one end. The requirement for pieces and components, their shape and functioning.

1.8. Applications of Vortex Tube

Vortex Tubes have a wide variety of applications and provide streams of

Cold air

- For cooling work-tool interfaces in machining operations and plastic injection moulds.
- For cooling parts in machining operations and electronic components.
- For cooling heat seals, environmental chambers and CCTV cameras.
- For setting of hot melts and cooling soldered parts.
- For cooling gas samples.

For air conditioning and for the cooling of drinks within the car, Vortex Tubes find their use within the automotive industry.

For various industrial applications like as freezers, cooling suits, CNC machines, Vortex Tube may be utilized.

In addition to the aforementioned cooling uses, the Vortex tube is also used as a hot pump to heat the space.

The Vortex Tube is a great instrument for refrigeration and heating gases, cleaning and drying gases, separating gas mixtures, liquefying natural gas etc. Where compactness, dependability and reduced equipment costs are key considerations.

1.9. History

By chance, a student named G.J. Ranque invented the vortex tube in 1932. Using a highly compressed input pressure source, he developed a pump that could generate two streams with different temperatures. A German scientist named Rudolf Hilsch improved the Ranque vortex chamber idea in 1945, and his results were published in an academic paper. The scientific community read Hilsch's paper and approved it, and the phenomena of energy separation in whirlpools became a source of great worry. He made a preliminary optimization of the vortex's tube shape. It's also possible to monitor the temperature difference between the input and exit streams. He also came to the conclusion that his innovative design, which lacked moving parts, was less successful than prior cooling systems. However, the mechanism behind this effect has remained a mystery. This device was later dubbed the Vortex tube Ranque-Hilsch, in honour of G.J. and Hilsch's contributions. The intricacy of operations in the vortex tube made mathematical or analytical analysis of the vortex tube very challenging. The flow was viscous, compressible, and highly irreversible. Mathematical modeling produced inconclusive results.

1.10. Theories And Methods

“To explain the temperature separation occurrences in the vortex tube, my team devised a number of theories. The transfer of tangential shear effort to exterior fluid layers, according to Hilsch[1], should be seen as generating energy separation. The main source of temperature separation in the tube, according to Ranque[2], is work transfer caused by compression and expansion processes. The evoked turbulent eddies, according to Harnett and Eckert[3], are caused by the process of temperature separation in the vortex tube. The turbulent motion has a wide range, and large eddies (associated with low frequency fluctuations) and small eddies (associated with higher frequency fluctuations) may coexist in the same volume of liquid. They claimed that a wide variety of eddies is to blame for temperature separations”. The mechanisms that generated energy separation were thought to represent Ahlborn and Groves'[4] secondary embedded circulation. “They measured the axial and azimut speeds to arrive at this conclusion. Stephan et al.[5] proposed the formation of vortex vortices on the inner wall of the vortex hopper as the cause of fluid motion in the vortex hopper. Secondary flows that flow through a concave wall in a restricted layer are known as Görtler vortexes. When the boundary layer thickness is equal to the curvature radius, the centrifuge action causes a pressure variation across the boundary layer”. This results in Görtler vortices and centrifugal instability in the boundary layer (Görtler instability).

Kurosaka [6] The acoustic streaming effect was identified as the source of temperature separation. Acoustic streaming is a continuous flow in a fluid produced by the absorption of high-amplitude acoustic vibrations. It's the sound that's the polar opposite of a flow sound.

“To conduct an experimental research, Scheper[8] used samples and visualization methods. The fundamental flow parameters were measured by the total temperature, speed and pressure. The axial and radial components of the speed were determined to be considerably less than the tangential speed. Takahama and Yokosawa [9] investigated the possibilities of reducing the length of the vortex tube by inserting a separate portion of the vortex chamber. In order to explore the phenomenon of energy separation, Eiamsa-ard and Promvonge [10] performed test experiments”. The technique utilized to introduce the air compressed into the vortex room was the snail entry. Due to the existence of a snail generator type placed at the nozzle entry, this technique was called the snail access method (see figure 2). They discovered that the cold side temperatures decrease improved and the efficiency of the vortex tube increased.

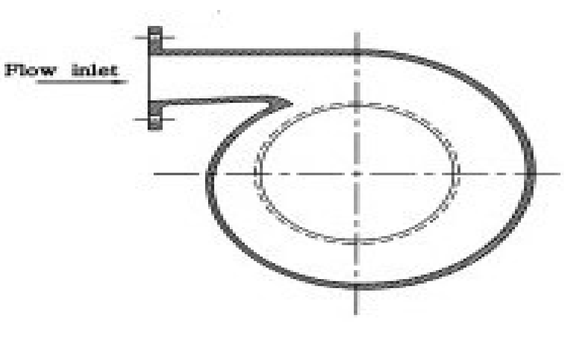


Figure 1.7: Snail type flow generator used at the inlet by Eiamsa-ard and Promvonge [10]

1.11. Applications Of Vortex Tube

Vortex is lightweight, compact and devoid of moving parts. It has no maintenance problems, because it does not utilize electricity or chemical substances. It is inexpensive compared to other cooling equipment, and unlike other localized cooling equipment, no coolant is required. She's resistant to wear since she's constructed of steel. Corrosion and oxidation are also resistant. The temperature is changeable, since both the cone valve and the generators may be changed by altering the user's geometric adjustment. It provides immediate chilly air, most essential. But it takes expensive compressed air to generate the immediate chilly air. Another key point to notice is that there is little efficiency in this separating process. However it is much overpowered by the benefits of the use of the vortex tube.

In view of these facts, localised cooling is increasingly gaining popular in the industries. It is utilised for cooling CCTV cameras and electrical controllers in the energy and electronic industries. The lens of a camera is also utilised for refreshing under intense radiant heat. It also serves to chill machine parts, refrigerate foodstuffs, and refresh the gas sample. In businesses where fast refrigeration is of utmost significance, such as the preparation of warm melts and soldered pieces. Other uses of vortex tube include cooling heat seals and environmental chambers. Even in certain "cryogenic," such as chilling laboratory samples and circuit testing, it is utilized. It is also utilized for machine/tools and deep mine refrigeration as well as ultrasonic weld refrigeration. Vortex tube is used to cool the ends of the plastic plate in the vacuum forming process to enhance the process performance.

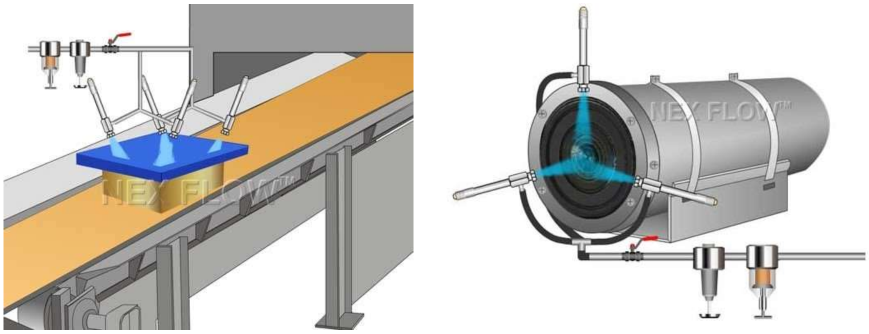


Figure 1.8: Applications of Vortex Tube: Vacuum forming process (a) Camera Lens cooling (b) Machine cooling

In aeronautical applications too, the vortex tube is utilized. An airborne contaminant may be removed from the engine bleed air by using a self-cleaning vortex tube. There are also proposals to power the Tesla Turbine using constant-flow exhaust gases from the vortex tube. The turbine is very long-lasting and extremely efficient. It should both resist the heat of the reaction.

CHAPTER 2

LITERATURE REVIEW

2.1. Literature Review

T. Dutta et. al. (2020) this article compares energy division into counter flow and uniflow vortex tubes for the same geometric and operational characteristics. This is an experiment that has been conducted. "There will also be a 3-D CFD study to determine the reason of differences in energy separation across various kinds of vortex tubes. The flow range is examined in both tubes and work and heat transfer rates are computed between the core and the perimeter. The tangential rushing work is recognized for the energy separation of the core region into the periphery in both counterflow and uniflow tubes, while the heat transfer from the periphery into the axial area decreases the energy separation".

Maheswaran et. al. (2018) Rankque Hitsh, "The Vortex tube absorbs the gas and it refreshes it. The variables contributing to this decrease include significant expansion, centrifugal strength, secondary circulation and friction. We have developed an air vortex tube for cooling and fluid operation. The measurements all contain temperature, pressure, discharge, weight, speed and other flux features. Our ideas were placed and gathered on the table. Input pressures were used: 7-6 bar, 6-5 bar, 5-4 bar and 3-4 bar. In the case of CNC machines, spot coolers and other applications, the Vortex tube is utilized as cooling device. Maintenance is quite low since this method does not have any moving components. The aim of this essay is to concentrate on two different geometric features of the vortex tube.

Deshmukh, et. al. (2018) This study investigates energy separation, radial variation, and maximum efficiency of the vortex tube using an exergy analysis to determine energy and flow phenomenon separation within the vortex tube. Air is a working fluid or coolant. Many million-dollar vortex tubes (1.2 million, 3.4 million, Valve Angle 300,450,600,900) with various geometrical features were used throughout the procedures. Cold mass accounts for between 0.1 and 0.9 percent of total mass. The exergy is measured using these factors. The Exergy has the highest efficiency with a 600 valve angle and a 0.4022 nozzle 4. In order to improve exergy efficiency with different working fluids, L/D ratios, hole diameter, cross-section tube, and valve angle form, further research on the vortex tube is required.

Kiran D. Devade et. al. (2016) Counter-flow heating and cooling are influenced by the cold end of the aperture diameters, the length-to-diameter ratio, and valve output angles. Working liquid for a Ranque-Hilsch vortex tube is air. Brass cold tubes and apertures were used in these tests. Experiments have demonstrated that increasing the cold tip diameter reduces the loss of exergy between heat and the cold fluid. Energy efficiency decreases when the L/D ratio rises. In a differentiated vortex tube, there is also evidence of reduced exercise loss than in a single tube. The loss of vortex exergy at the hot end is influenced by valve angles.

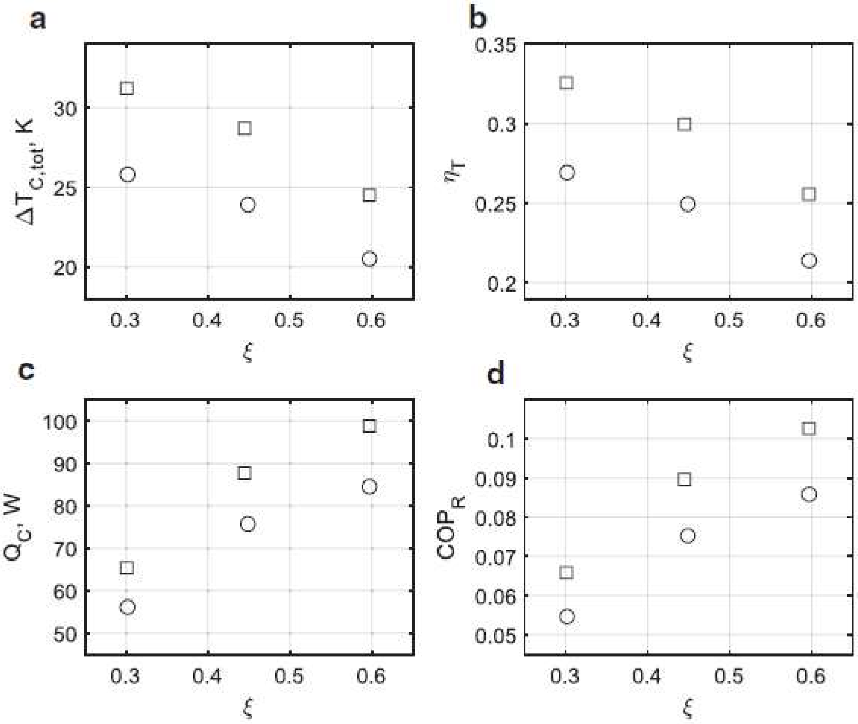
K. Dincer et. al. (2011) In this study, three Ranquee Hilsch vortex tubes with an inner diameter of 9 mm and a length-to-diameter ratio of 15 were used. Their performance was divided into two categories: hot cascading type RHVT and traditional RHVT type RHVT. Performance analysis is difficult due to the temperature differential between the heated exit and the intake (D Thot.). The Ranquee Hilsch vortex values in the hot-cascade tube were greater than RHVT's D Thot value. The total input exergy, total output exergy, total exergy loss, and efficacy of the hot stream were calculated using experimental data. The total loss of power as a proportion of the cold flow reduced in both conventional RHVT and hot waterfalls. It has also been shown that RHVT is more effective than traditional RHVT heat cascades.

Seyed Ehsan Rafiee et. al. (2013) Experimental research is carried out on the dust, intake pressure, and number of intake convergence features. It will study the impact of 1e2.85 nozzle convergence ratios. The main aim of the research is to show how CFD can be used to build a tool that can be used with confidence in a broad array of conditions and geometries and therefore provide a strong tool to optimise the design of vortex tube and assess its use in new applications. The performance of the vortex tube system was predicted by development of a computational dynamics model. FLUENT 6.3.26 was utilised to carry out a complete 3D (3D) CFD simulation. This model uses the key 3 turbulence model to solve flow equations. Experiments to validate the findings of the simulation have also been performed. The first objective of numerical research was to validate these findings with experimental information and the second objective was to optimise the experimental model for the highest possible performances.

Bazgir et. al. (2019) Using different turbulence models (the temperature drop of the cold air (Tc=Ti) within the Ranque-Hilsch Vortex tube direct counterflow) in the Ranque-Hilsch Vortex tube direct counterflow, this article demonstrates how temperature separation may be influenced (RHVT). The method is used to create an optimum turbulence model for energy separation, which is then compared to the experimental data collected for the system. Several turbulence models have been investigated in both steady and transient time-dependence modes in order to get results with the least amount of error. The findings show the maximum consistency between the RNG k-p turbulence model and the experimental data obtained from the setup; as a consequence, the RNG k-н model for Finite Volume Method was used for all computations (FVM). Once a specific length of hot tube has been exceeded, the cooling capacity of the hot tube is affected considerably more.

Manickam et. al. (2019) The vortex tube or Ranque–Hilsch vortex tube is a device for separating hot and cool air into the vortex chamber via the input boxes through the tangential flow of compressed air and examining the power separating effects of a vortex tube using a dynamics model for computing fluids. When standard k-epsilon was employed, the control equations were solved in ANSYS fluent code, which were compressible and turbulent in 3D. There were investigations on the impact of geometrical factors. The flow behavior of the vortex tube has been calculated numerically. The three-dimensional vortex model with static condition is created using this method.

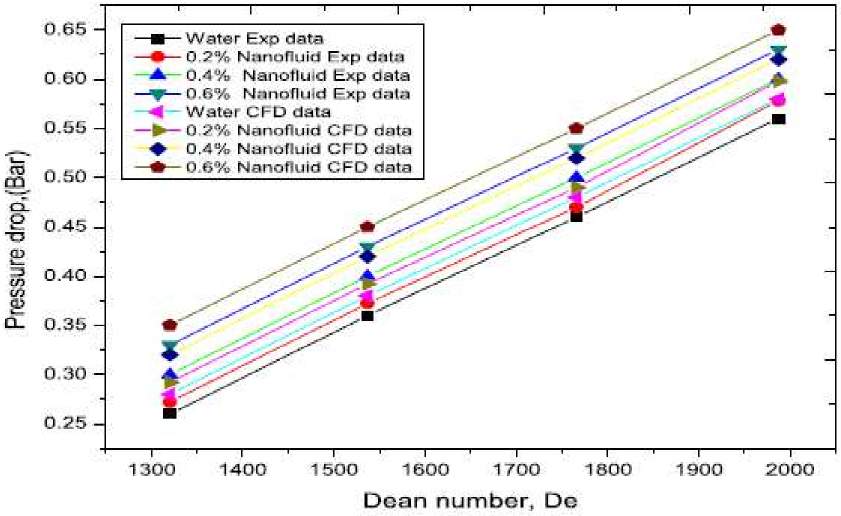
Matveev et. al. (2019) Increasing vortex tubes with cyclonic extensions of the vortex chambers were numerically explored. A standard air-flow vortex tube was used in the computational fluid dynamics programme STAR-CCM +. Compared with experimental research found in literature, the numerical method was verified. The parametric analysis was carried out on a fairly effective high-capacity vortex tube sign. Also across a range of cold-flow fractions, this version of the vortex tube was expected to be more effective than the original system by approximately 15%. An intermediate size vortex-chamber extension has been discovered to increase cooling capacity while producing the lowest temperature at the cold outlet. A consistent improvement in performance metrics in the studied cold-flow percentage was achieved compared to the initial configuration without a vortex chamber by about 15%”.



“Vortex-tube version 1 (circles) and variant 4 (squares) performance metrics at various cold-flow fractions.

Mukesh Kumar et. al. (2019) The computer programme ANSYS 14.5 has analysed the heat transfer and the pressure reduction of the dual-helix heat exchanger handle MWCNT/water nanofluids. In the Dean range of 1300-2200, the computational analysis was performed in the laminar flow condition. A conventional method was used to design the new heat exchanger shell and double-helical tube tube and 3D models were performed in Cre-O 2.0 parametric. The ANSYS Workbench 14.5 Finite Element Analysis software was utilised for typical workplace CFD analysis. For this research, the MWCNT/water nanofluids were collected with 0.2%, 0.4% and 0.6% with volume. CFD study under the normal operating state was carried out using the ANSYS Workbench 14.5 Finite Element Analysis programme. For this study, 0.2% of the MWCNT/water nanofluids, 0.4% and 0.6% volume concentrations have been collected.

The heat transmission rate and pressure decrease were investigated as the volume of MWCNT/water nanofluids increased. The number of MWCNTs/water nanofluids in Nusselt is found to be 30 per cent greater than that in Dean at 1,400 and pressure drops at Dean at 2 200 are 11 per cent higher than that in water. There is a fair deal of agreement with the experimental data in the simulation data.



CFD pressure drop Vs experimental pressure drop.

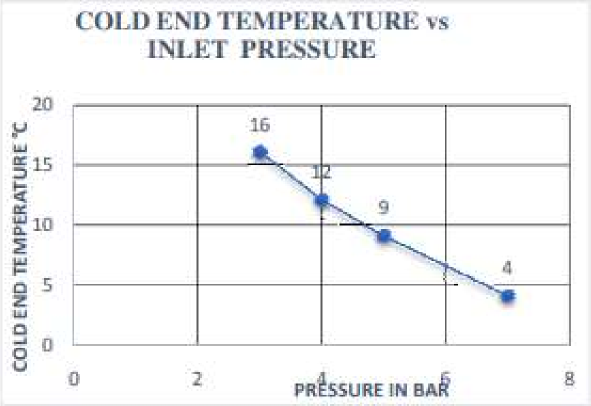
Avijit Nayak et. al. (2019) The goal of the CFD research presented in this paper is to uncover previously unknown features of vortex tube performance. The mechanism of flux and energy separation is investigated and described. Using the data provided by the researchers, the computational technique was tested and confirmed. Parametric studies are carried out with various nozzle counts, inlet pressure, inlet temperatures, ratio of hot end tube length-to-diameter, cold orifice diameter, nozzle diameter, and so on. The vortex tube's isentropic efficiency was calculated using CFD data, cold mass fractions, and the coefficient of performance (COP).

Madhu Kumar et. al. (2019) In this research, an investigation was undertaken on the basis of the findings of CFD on the appearance of the two-stage hot vortex tube with a diameter (L/D) length of 10 and 15 with various pressures of 8, 9, 10, 11 and 12 bar. A CFD programme data were used to examine the temperature variation between the hot inlet and outlet, the cold outlet and the inlet (Tcold). The difference between the high temperature outlets and the inlet (aliasThot), the cold outlet and the inlet (aliasTcold) was determined using an L/D ratio of 10 and an inlet air pressure of 12 bars.

Ms. Shreya Sawaibahadure et.al. (2019) The vortex tube is a device that distributes hot and cold air to both ends by boosting the pressure air to the appropriate level. This is one of the best examples of temperature fluid and pressure changes that I've ever seen. The different components may be arranged in numerous ways to alter the temperature on both ends. The fluxes that cause the temperature fluctuation must also be investigated in order to fully understand the temperature variance. Various techniques are used to assess the thermal performance of vortex tubes. There is an emphasis on the flow and thermal performance of existing research. There has also been some study that focuses on exergy analyses. The flow characteristics are also important, and the effect of the flow on temperature change must be investigated. This page discusses different studies on vortex tubes and methods for thermal efficiency assessment. Additional suggestions for better thermal performance and flow have been provided as a result of the study.

Domenico et. al. (2018) This research demonstrates an efficient method that is based on common commercial instruments. The Ansys mechanical FEM solver utilised to extract the particular modes of its own. The modal forms are then imports fluently into the cfd solver using RBF morph add. The modal coordinates may be updated and the model form can be adjusted by taking account of the elasticity of the cfd model. Temporary analysis is confronted with a solution to time by altering the mesh form in each phase".

Maheswaran A et.al. (2018) Hitsh Rankkaque "The Vortex tube is used for the production of cold gas. Sudden expansions, centrifugal action, secondary circulation and friction are the main causes ascribed to this decrease in temperature. A vortex tube has been manufactured which utilizes air as work fluid and has a cooling effect. Measured flow properties like temperature, pressure, discharge, volume, speed etc. We tabled and drew out our observations. For the pressure of 7-6bar, 6-5bar, 5-4 bar and 3-4bar the values have been obtained. In CNC machines, spot cooling applications etc., the vortex tube is utilized as cooling device. This method needs minimal maintenance since no moving components are involved. This article is aimed in general at concentrating on two aspects: geometrical properties of the vortex tube and the associated thermophysical parameters".

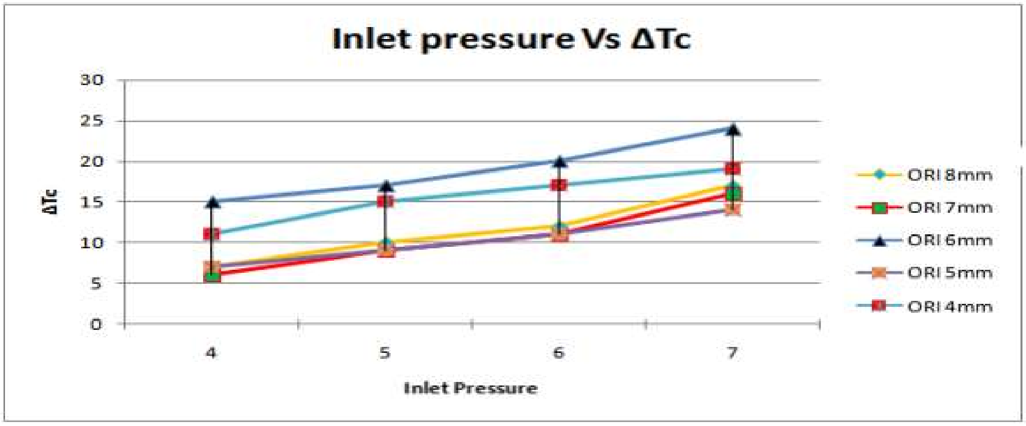


R. Dhineshkumar et.al. (2018) In several nations, sustainability is established; especially in the area of engineering, production and supply chain management. Sustainable means that the requirements of the current generation are not met by taking the need to manufacture features. “The rising desire for a more eco-friendly society and an expanding population is making sustainability increasingly essential in today's culture. Environmental, social and economic reconciliation is sustainability. Vortex tube is being utilised for the production system as a sustainable tool. Vortex tubes without moving component thus no maintenance is necessary. This study consists of numerical analysis on improving the vortex tube parameter to minimise the cold temperature. Use the numerical tool to tune the length and diameter of the vortex pipeline, cold orifice radios and hot end gap. Also changed and analysed the intake profile of the vortex tube. It is important to reduce the cold end of the vortex tube temperature. In machining alternatives for the flood refrigeration technique, we may utilise the vortex tube in the machining process.

Pavan R. Bangare et.al. (2018)A vortex tube is a simple, moving device that uses compressed air flow to produce cold air in the cold end and hot air in the warm end. The air spirally travels around the heated side perimeter of a vortex chamber, creating vortex movement. The tube's air vortex has a central and exterior zone, lowering the core zone temperature while increasing the external region. This study presents the results of a variety of experiments focusing on different hot-end geometries for various intake ports. Three vortex tubes will be used in the design, manufacturing, and testing of the greatest temperature decreases with 1, 2, and 4 intake ports. The behaviour of a vortex tube system is investigated via an experimental test. A reliable test equipment must be used to determine the effect of the inlet number port.

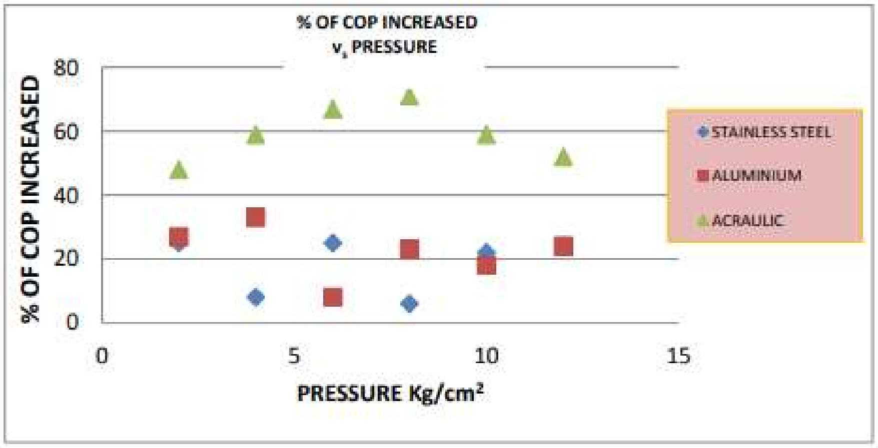
Ramesh ganugapenta et.al. (2018) In preparing countries, refrigeration has a significant role in protecting nutrition, prescription and aeration and chilling in particular. Freon is used as a coolant in normal cooling frameworks. Moreover, the possible cause for ozone drainage is, wide study is under way into frameworks of exchange cooling. Vortex tube is a non-common cooling tool, which does not have moving components that generate chilly air and warm air from the compressed air well without harming nature. When high weight air flows into the vortex chamber inventively, a brawny vortex stream is moulded, which is divided into two air streams. The main focus of the current study is on the development and production of the Delrin material vortex container. The execution of the Vortex tube is measured for different opening and canal weight measurements after production. Cooling and thermal effect and

COP as implementing methods are selected. The following are the major goals of the current work: Determining the appropriate hole diameter for the impact of cooling and heating. To determine the appropriate intake pressure for the cooling and heating effects. Validating the optimum diameter for selected COP performance metrics.



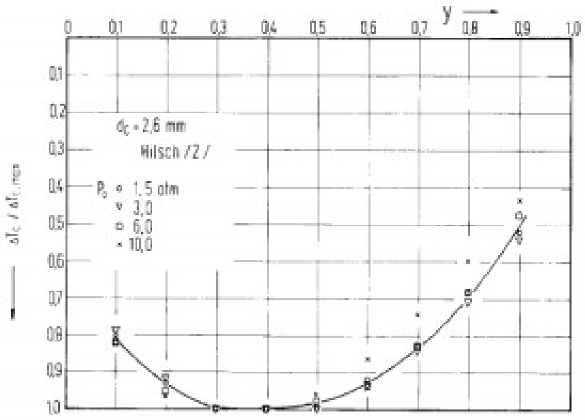
The above shows the effect of orifice diameter and pressure on the ΔTc.

K. Kiran Kumar Rao et.al. (2018) Cooling plays a vital function in human existence, influencing the human field, namely the sustainability of food grains and medicines, the enormous storage of chemical products and medicines, industrial cooling and air conditioning. The generally used cooling system utilises Freon as a coolant that pollutes, lowers the ozone layer and poses a health risk. Proxy cooling techniques are used to avoid this damage by significant research effort. Vortex tube is an easy-to-use gadget, a non-orthodox freezing device capable of separate hot and cold air from a requested intake air stream without altering the terrain. Due to its small weight, modesty and efficiency, this gadget is essential for applications.



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

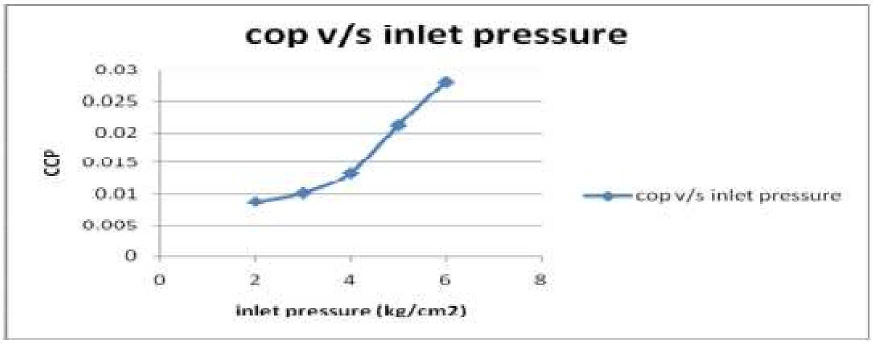
Sayali Darekar et.al. (2018)A vortex tube is a simple device that divides a hot and cold gas stream into a heated stream without chemical or external energy. without any chemical interactions. It is a mechanical device with no moving components. The splitting of the flow into low- and high-temperature zones is called the temperature separation effect. Two fundamental parameters depend on the performance of the vortex tube. First, the working parameter such as inlet pressure of the compressed air; and second, the geometrical parameters such as number of nozzles, nozzle diameter, valve angle of the cone, hot side tube length, cold orifice diameter, and vortex tube material. The Vortex tube has fascinating functionalities and many industrial applications and is employed as a cooling unit in industry as a refrigerator".



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

Mr. Shadab Shaikh Bismillah et.al. (2018) Today, the ecological nature of any research or development is the most important characteristic. "As we know it, safety has become an important component of business and bringing people together. As we pass through compressed air from a compressor, the vortex tube is also known to produce hot and cold air, which is not detrimental to the environment. The compressed air in the vortex tube is split into two parts by tanks that are tangentially translated into the vortex chamber vortex tube, resulting in a free vortex as a peripheral warm stream at the conical vent end and a forced vortex as an interior cold stream through the orifice. It may be used for any kind of refreshment on the go.

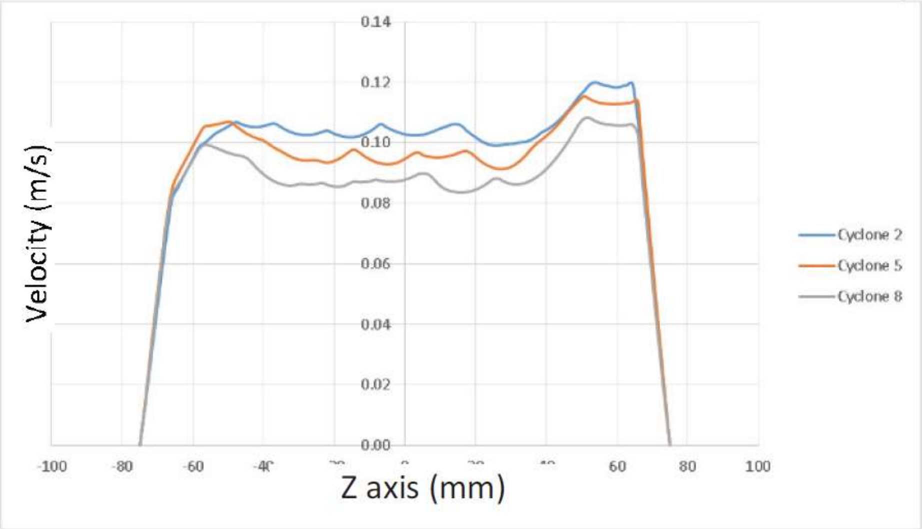
Anil Kumar Bodukuri (2017) Vortex tube is a mechanical system that does not include moveable components. The transmission of compressed air leads to the generation of two extreme temperature air streams. This unique feature of the vortex tube is utilised for many applications such as heating and cooling. Vortex tube simultaneously generates hot and cold air from both ends. The primary emphasis in this article is on the low temperatures that create the cooling effect. Four various tubes are manufactured and their POOs are compared, with the number of nozzles, apertures & ventures.



COP Vs Inlet Pressure.

Hamdan et. al. (2017) This study uses numerical analysis to better understand the flow field and temperature separation in a Ranque-Hilsch vortex tube. Using ideal gas compressible flow assumptions, a three-dimensional computational dynamic fluid model is utilised to evaluate the performance of a vortex tube. The research shows that the behaviour of energy separation may be predicted numerically using the ideal gas assumption. With modified vortex tube dimensions, the numerical study shows the maximum temperature separation at an optimum pressure value of 4 bar. When using isolated tubes, the energy separation increases as the tube length increases until an asymptote value is reached. When the gap is at its maximum value, there is an optimum diameter.

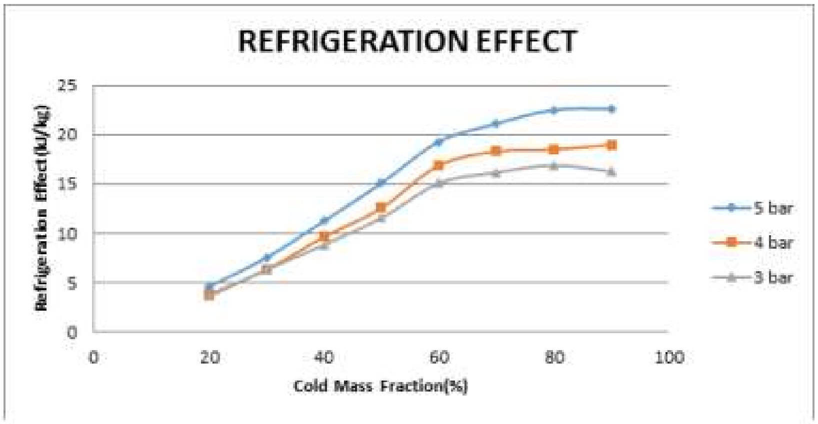
Surjosatyo et. al. (2017) ANSYS Fluent was carried out as an experimental investigation and a CFD simulation. The experimental research is to detect vortex finder dimensional impacts on the efficiency trend of collection, while the objective of CFD simulation is to estimate the air speed profile inside the cyclone separator. RNG k-β was selected as the turbulence model for swirl dominated flow. The cyclone separator uses speed magnitude to forecast the trend of collection efficiency. The weighing technique, i.e. a percentage of the total mass of ash collected and a total mass of ash entered the cyclone separator are used to calculate the collection efficiency from the experiment. Results from experiments indicate that increased length of vortex finder and reduced diameter of vortex finder may improve collecting efficiency. In general, extending the length of the vortex finder and reducing the diameter of the vortex finder may also help to speed the air in the cyclone separator. Use of a whirlwind Finder with a diameter of 35 mm and a length of 1 D (150 mm) to achieve maximum collecting efficiencies.



Velocity chart at vortex finder length 1 D.

Xiaojie Zhai (2017) The problem of heat damage in high-temperature mines is becoming more well recognised. Furthermore, existing systems have large initial expenditures and poor performance in terms of safety. In this article, the idea of vortex tube cooling is applied to a mine cooling jacket kind of Vortex tube. It has used compressed exhaust gas as a gas source to solve the problem of large-scale electrically supported cooling equipment without releasing dangerous gases at the same time. Using the observation method and a questionnaire survey, the parameters of the vortex tube were modified for the best cooling effect. The vortex robe kind of cooling jacket may also provide security, dependability, energy efficiency, and environmental protection. We also looked at how the vortex tube kind of mine cooling jacket might be used.

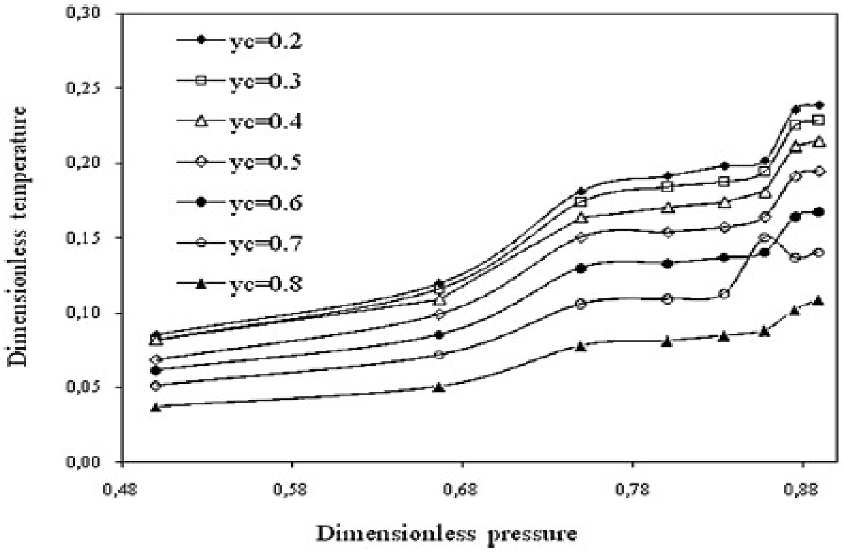
R C Venkatesh et.al. (2017) This article describes the performance of the vortex tube depending on several geometric factors such as pressure, gas, L/D ratio, and cold weight fractions. The experimental outcome was the cold-end temperature decrease. With a 1 bar increase from 3 to 5 bar inlet pressure, the cold weight percentage was similarly between 20% and 90%. The maximum cold weight fraction was 60%. At the L/D ratio at 17.5 and at pressure 4 bar maximum cold mass fractions were achieved. When testing different component parts such as air, carbon-dioxide and nitrogen gases, at a ratio of L/D to 17.5 a maximum cold weight fraction of Co2 gas was achieved. The maximal intake pressure for air, nitrogen, oxygen and argon occurred with a maximum temperature gradient of 700 KPa. For argonaut 52.1o C, the maximum temperature difference was achieved. With increase in nozzle number, the temperature gradient reduces. At nozzle number the maximum temperature difference was 2. Finally, the performance changes according to the temperature level with various parameters of the vortex tube.



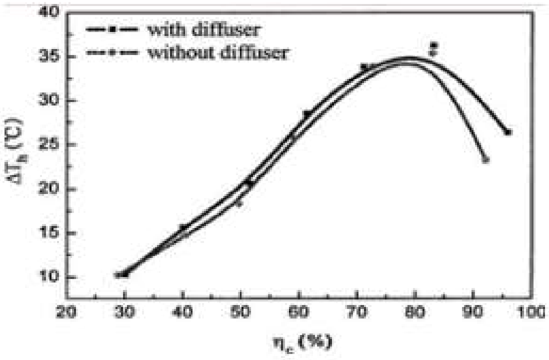
Upendra Sharan Gupta et.al. (2017) The vortex tube was discovered by Ranque, and Hilsch was the first to describe it in detail. Commercially, vortex tubes are being used for low-temperature applications, such as cooling machinery components, setting solders, chilling electronic control cabinets, cooling food, tempering sensors, and dehumidification gasses Vortex tube is a simple device with no moving parts. At one end, it splits the high-pressure air intake into two cooler, lower-pressure streams, while at the other end; it divides it into two hotter streams. The flow type and the counterflow type are almost identical. It has a number of advantages and applications, the majority of which are more efficient than uni flow counter flow. The majority of applications emphasise performance, energy efficiency, compactness, or an alternative to the traditional method. Vortex air coolers are suggested for many industrial applications. A vortex tube in the personal air-suit enables workers to work longer hours in difficult conditions. Despite improvements in tool materials, contemporary tool tips can presently maintain their cutting edge at greater temperatures; the cutting edge will eventually break down. The application of cold air on the tool interface of these modern tool tips will extend tool life and reduce metal cutting costs".

Mr. H R Ghan et.al. (2017) Vortex tubes are an air cooling technique, and since air is abundant in nature, vortex tubes are both cost-effective and environmentally hazardous. "The vortex tube is a mechanically simple device with no moving parts that may split a high-pressure flow into two lower-pressure flows with different energies, which are typically represented as temperature differences. Several design techniques are used in the construction of the vortex tube. The characteristics of heat transmission in a whirlwind tube are investigated in relation to various parameters such as cold/hot end cross-sections, compressed air input nozzle area, cold orifices, tube hot end, and L/D ratio in a literary research. There is no such thing as a perfect theory, thus various researchers may come up with plausible explanations for the vortex tube phenomenon. As a result, this article examines the different design techniques and analysis that were intended for testing".

Celik et. al. (2016)"In this research, a basic information and the impact of different design parameters on the features of tubes performance was produced via the setting up of an experimental vortex tube system to examine the predefined parameters for the design of vortex tubes and thermodynamic analysis. In this research an experimental configuration was constructed and several design parameters (Pi, d/D, AN, yc and L/D) were investigated to allow for optimal conditions of this kind of tubes. In addition, the COP of vortex tubes and traditional methods for heating and cooling were compared. In the main, the vortex may be utilised in locations with compressed air sources and the pressure must be reduced in circumstances".



Ankita Adana et.al. (2016) Technology spreads and pollutes our environment by emitting harmful components that cause ozone depletion and make people uncomfortable. “The thesis' goal is to look into the concept of a vortex tube that can be used to replace conventional cooling methods for refrigeration now and in the future. The research to enhance the cooling effect by changing the tube geometry, changing the cylindrical shape of the tube to a tube at various cone angles, installing a diffuser between the tube and the hot tube, and why the tube is separated from the vortex by energy is discussed in this article.

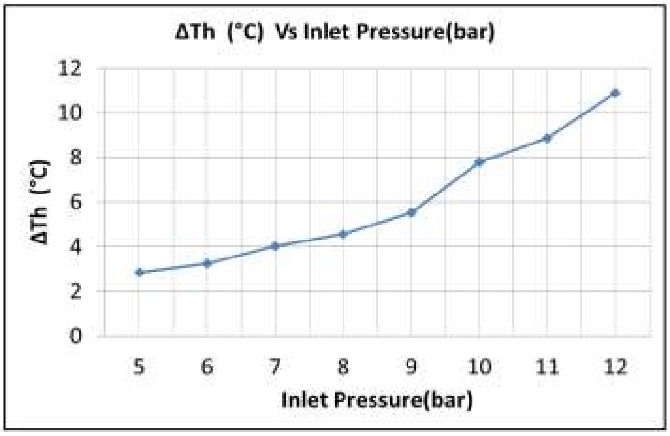


Effect of the diffuser on heating effect.

Krishna Kumar Karothiya et.al. (2016) We have built a vortex cooling system in this article. Effects of the tank intake, tube diameter and nozzle diameter were studied experimentally for the temperature decrease in the tube. This thesis is motivated generally by clarification of many assumptions and optimum data collected from different study articles to evaluate the resulting temperature difference. Standard measurements and manufacturing methods have been upgraded to get reliable findings compared to prior research. Two new methods are used to acquire new measurements: I Nozzle for tangential air flow of the compressed air is installed. ii) isolated heat loss possibilities are removed. (ii) The cold air temperature decreases and compares the separation circumstances at a comparable level. The test findings indicated that the vortex tube with tangent intake nozzles and 12, 7 mm cold tube diameter provided the greatest 6^0C decrease.

Dhillon et. al. (2015) In comparison with the experimental findings accessible in literature, the impact on VT is investigated in the modification of the angle of difference of the hot tube. In both a straight and 2 degrees flared pipe, the temperature separation between heat and cold output is investigated. Analysis findings reveal that the flared tube is more cold than the straight tubing for a hot mass fraction over 0.5. Effect on thermal transfer and shear work transfer in the VT have been studied of key factors such as temperature gradients, vehement (axial, radian, tangential), speed gradients, effectiveness of thermal conductivity, viscosity of fluid etc. The axial direction heat transfer and work transmission in both straight and flared tubes were studied in order to understand the temperature separation process. The efficiency isentropic and COP as a cooler as well as a heat pump with a tube and flared tube were calculated.

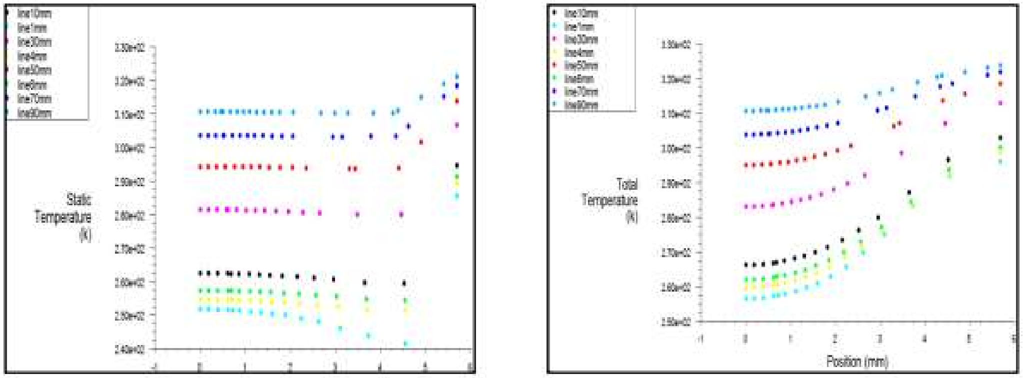
Sagar N. Jadhav et.al. (2015) The vortex tube cooling setup was studied experimentally. As a counterflow tube, the vortex tube was designed and produced. A vortex tube is a basic mechanical device that may be used for cooling and heating. It works using compressed gas that is split into two streams of hot and cold gas at opposite ends. Air is used as a fluid in this instance. The performance of the Ranque-Hilsch vortex tube is influenced by a variety of variables (RHVT). These parameters may be divided into two categories: working parameters and geometric parameters. Compressed-air input pressure, cold-weight percentage, ambient air temperature, and geometric settings such as bucket diameters, warm and cold side, heat-and-cold side lengths, and conical valve angle are all working parameters. The effect of these variables is discussed in this research. Because of its poor heat conductivity, UPVC is used as a pipe material. Unplasticized polyvinyl chloride is the material. The performance of the vortex tube is examined and investigated in this article.



Increase in ΔTh with increase in inlet pressure Pi.

Alok Manas Dubey et.al. (2015) This article provides a theoretical study on the thermal performance of a natural cascade cooling-heat pump system for heating and cooling. For carbon dioxide-propylene, a parametrical assessment is also given using other factors such as evaporation, the exit temperature of a gas cooler and the temperature difference in a cascade heat exchanger system. In place of an expansion valve the overhead cycle of the system is supplied with a vortex tube. Compared with the transcortical cascade cycle without the vortex tube expander in the supposed operating parameter range, the greatest improvement in the current system performance is 5.9 percent. The model forecasts performance characteristics and operation circumstances at several stages such as pressure, enthalpy, temperature and entropy. With a multi-linear regression analysis, the suggested transcortical model is optimised for total performance coefficient, intermediate temperature and mass flow ratio. In tables and figures are shown the effects of the major factors on optimal design quantities. A coefficient of performance values is compared with various reference cascade systems to verify a cascade refrigeration and heat pump model, using carbon dioxide/propylene as refrigerants. This research may contribute to the future development of carbon dioxide-propylene cooling systems and their optimum design.

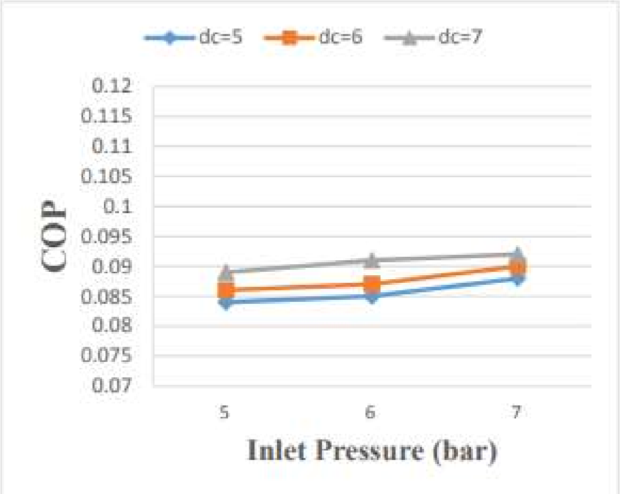
Hitesh R. Thakare et.al. (2015) This article concerns CFD research of the flow vortex tube counter. The CFD is a 3D model that uses the conventional k-epsilon turbulence model. In this CFD research, fluid behaviour in the vortex tube has been studied. The goal of this study is to demonstrate the effective use of CFD in that context and to provide a strong tool to optimize and evaluate the value of vortex tube design in new applications. The findings of this CFD research accord well with the results of the experiment.



Static Temperature inside Vortex Tube and Total Temperature Inside Vortex Tube.

Shrikrushna Nagane et.al. (2015) Cooling is critical for both humans and machines. In general, cooling methods such as vapour compression cooling and vapour absorption cooling are used. The cooling system of the Vortex tube, which is not widely used for refrigeration, is an example of an unconventional cooling system. The conventional colouring technique has many advantages in the Vortex tube. Cold and hot streams may be divided using vortex tubes in industrial applications such as coolers and coolants. The cold and hot end cross sections, compressed air nozzle input area, cold aperture area, and hot end section of the tube are all essential features of the vortex tube.

Akash S. Bidwaik et.al. (2015) The focus is currently on designing a turret tube with a diameter of 5, 6, 7 mm. To increase the intensity of the turbofilm, the test uses an eight Tangential Nozzle L/D = 11.5 with a double intake turbocharging tube. The experimental data and analytical values were meticulously recorded and presented in the paper in a clear and concise way. To assess the fluid's performance within the vortex tube, a numerical analysis was conducted utilizing different geometrical and thermal physical factors. The regulatory equations were solved using ANSYS FLUENT 15.0 and a 3D model on a fluid domain. The design proved to be a model for others to follow.



Effect of inlet pressure variation on COP for twin entrance vortex tube with varying diameter of cold aperture.

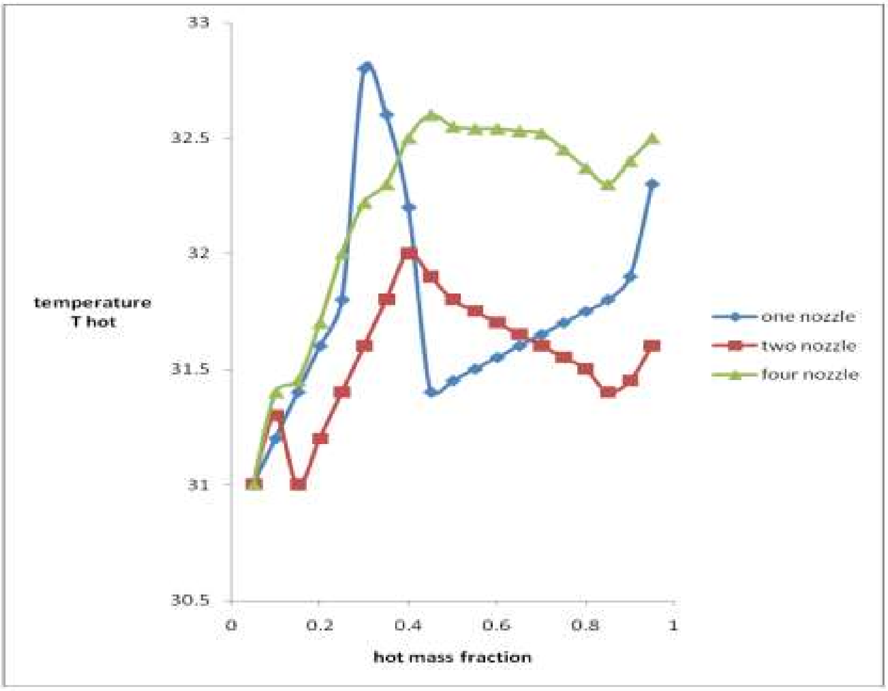
K.K.Arun et.al. (2014) The future of sustainable development is innovative methods that minimize the effect of production on the environment. The growing desire for a more eco-friendly populace makes sustainability increasingly essential in today's society. Vortex tube is a sustainable spot-cooling gadget that requires no coolant or power. Vortex tubes are free of maintenance and no maintenance is needed. This study optimizes the cooling temperature parameter of the tube. Different CFD analysis vortex tube of different length. The CFD study provides the optimal cold end (dc) diameter, the length to diameter ratio (L/D) and optimal parameters to achieve the lowest cold gas temperature. The significance of this study is in showing the vortex tube as an alternative and sustainable spot cooling technique for machining operations. The efficiency of vortex tube with various profiles, lenght and diameter ratios is analyzed using the CFD tool".

Prof. Uday V. Aswalekar et.al. (2014) The Ranque's Hilsch (Ranque's Hilsch) The Vortex tube is a mechanical device that creates a localised cooling effect by separating pressurised air into hot and cold streams. "It is used in a variety of applications because to its simplicity, sturdiness, and ease of maintenance. Its design is still based on empirical connections and rules of thumb from experiments. Because of the complexities of the stream, it is difficult to capture the whole flow and energy separation properties of vortex tubes. The objective of the numerical analysis of the vortex tube is to reveal previously unknown features. An L/d ratio 10 vortex tube with two straight, circular intersecting nozzles is used in this study. Solid Works 2010 is used to create a three-dimensional cavity model. The flow, press distribution, and temperature distribution along the length of the vortex tube are all evaluated using ANSYS FLUENT 13.0. The route lines for vortex tubes fit the theoretically stated course.

Abu Bakar IZHARa et.al. (2014)ANSYS CFX simulation code is used to model vortex-inductive vibrations of a cylinder in this study. The cylinder is regarded as a rigid body and the employment of a one-degree freedom spring damper system results in cross displacements. Air as a fluid is used for both 2-D and 3-D analyses. The number of Reynolds, which covers laminar and turbulent flow regimes is variable between 40 and 16000 roughly. For validation use are made of the experimental findings (Khalak and Williamson 1997) and other researchers. Comparable outcomes are achieved.

Nilotpala Bej et.al. (2014) Ranque-Hilsch vortex tube (RHVT) is a simple tool which divides a compressed input gas stream into a cold and hot output stream without an additional energy source. The hot gas stream coming from the first stage vortex tube is provided in RHVT type of hot cascade at the entrance of the second stage vortex tube and thus produces a more heated effect. This article provides the findings of an exergy analysis for second stage RHVT for various fractions in cold, using the standard keć turbulence model. The findings obtained from numerical simulations are favourable for the application of the RHVT cascade model for the experimental measures available.

Sarath Sasi et.al. (2014) The most essential good feature in the refrigeration industry, which is connected with nearly any analyses or even advances, is their green mother nature, which satisfies your fundamental requirements without damaging the environment. These businesses and people are now working together to ensure environmental security. The goal of this article is to improve the effectiveness of just one green technique for commercial cooling, known as vortex tube, and to refresh process requirements such as location cooling, solder cooling, inexpensive slitting, extrusion cooling, and so on. Using these commonly used cooling methods, the fuel works with fluids that either degrade the ozone layer or contribute to increasing global temperatures in accordance with CO2. Many features have previously been included in attempts to achieve optimum production using knowledge of this vortex tube. I'm looking at a vortex tube for the purpose of constructing the most cost-effective tube possible. To that end, I created an optimal vortex tube and conducted experimental research on the vortex tube by changing various variable numbers, turbo materials, various cone angles, and various mass fractions.

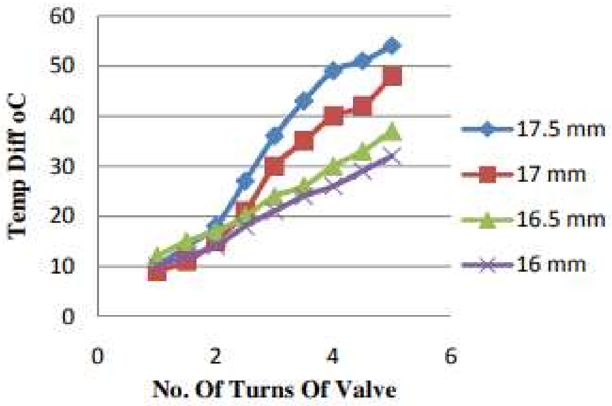


Hot end temperature v/s hot mass fraction.

Waraporn Rattanongphisata et.al. (2014)A vortex Tube is a cooling device where an air may also be utilised as an ecologically beneficial working medium. Studies of thermal separation flow and early testing indicate the potential to improve vortex cooling capacities by lowering the temperature of an external surface at a hot tube part. A thermoelectric module is used here to extract heat and release it into the environment from the surface of the hot tube. In other words, the thermoelectric generator as thermal energy harvester uses a tempering difference. The experimental plant on the vortex cooling system, which includes the thermoelectric module, is developed and built in a laboratory. Cold fraction from 0 to 1 and air inlet pressure of 1.5 bar constitute the test conditions. The findings indicate that when the thermoelectric module extracts heat from the hot tube surface of the vortex tube and electricity is produced as per product, the cooling capacity and effectiveness are increased. The cooling power of the vortex tube and its efficiency improved by 4.3% and 9.6% respectively.

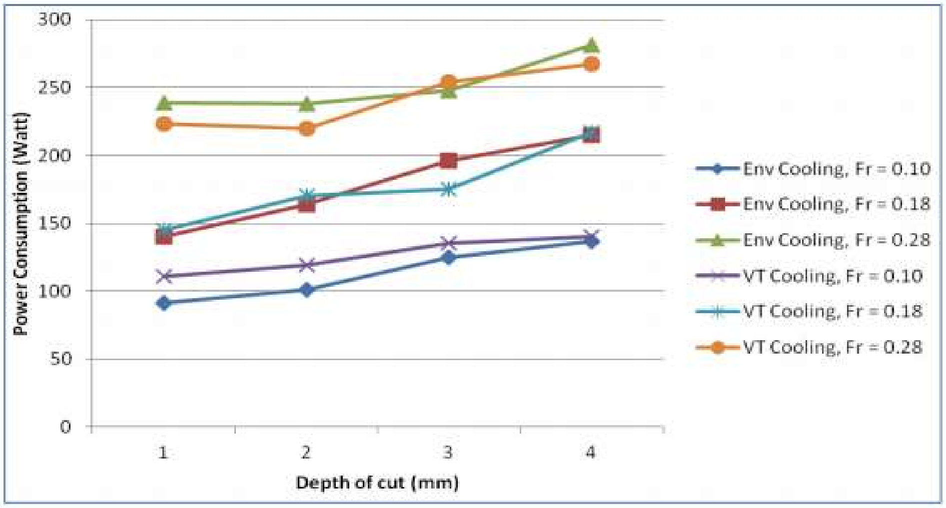
Anatoliy Khait et.al. (2013)"Numerical simulations utilising various Turbulence Models visualise the air flow in the Ranque-Hilsch vortex tube. During calculations: k-β, k-µ realisable, k-µ RNG, SST and SAS-SST, the following turbulence models were utilised. It was discovered that the presence of massive secondary vortex structures in the computational domain is only predictable in the SAS-SST turbulence model. Different experimental investigations indicate the presence of massive secondary vortex formations".

Sankar Ram T. et.al. (2013) The Ranque-Hilsch effect is a temperature distribution phenomenon that occurs in confined, constantly rotating gas flows. The basic counterflow vortex tube is a long hollow cylinder with a compressed air injection tangential nozzle on one end. The flight within the vortex tube of a spring-formed vortex track may be described as rotating air. The peripheral flow is directed towards the heat end by a heat end plug, while the axial flow, which is forced back by the connection, is directed in the opposite direction to the cold end.



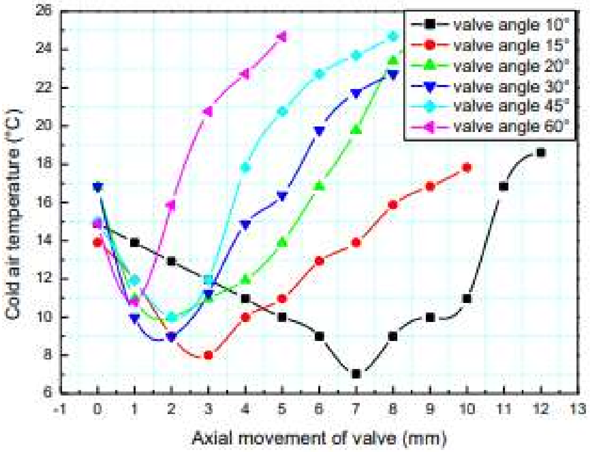
"Temperature Difference vs Mass Flow Rate (For Diffrent Hot Plug Openings)".

Zahari Taha et.al. (2013) We have grown more aware of the need to reduce environmental impacts over the last 10 years. The manufacturing industry aims to minimise environmental impacts via greater features, less processing power and less coolant uses for new materials. The main objective of this study is to assess the effect on energy consumption and quality of surface ruggedness of the vortex cooling tube from Ranque–Hilsch while utilising the carbide-recording tool used to manufacture softer steel. The machining parameters involved in the experiment are cutting speed, feed rate and cutting depth. The cutting speed is 160 m/min and the feed rate and cutting depth vary. The feeding rate must be 0.10, 0.18 and 0.28 mm/rev, and the cutting depth shall be between 1.0 and 4.0 mm. Infraround thermometers are used to monitor the cutting temperature throughout the machining process and a power and harmonics analyzer is utilised to measure power usage. The machined components are measured using a robust surface testing equipment. Results show that Ranque–Hilsch vortex cooling reduces the cutting temperature, but under environmental conditions power consumption and surface ruggedness is more than 0.28 mm/rev.



"Power consumption performance with respect to different depths of cut, feed rates and cooling methods".

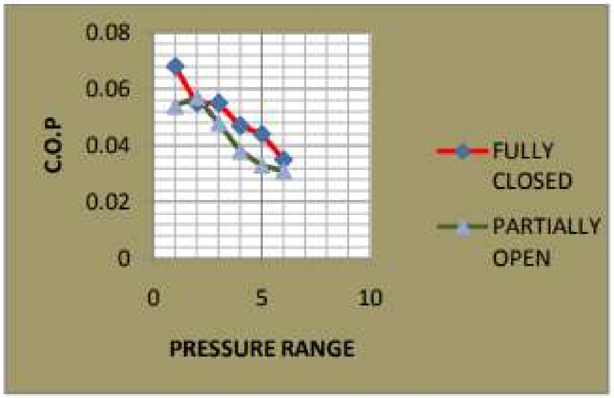
S.Rejin et.al. (2012) This article covers the experimental research on the vortex tube fridge of the cold side and on the performances of the vortex tube fridge with a variable conical valve angle on the hot side. The project began with the design and manufacture of a vortex tube fridge. At the input compressed air was inserted at the pressure of 5bar. A maximum temperature decrease of 7°C was obtained for these specified intake circumstances. This result achieved a conical valve angle of 10° and a cold orifice diameter of 6mm in the working state. The results of the test indicate a temperature fluctuation with different conical angles and a diameter of 6 mm at the lowest temperature. The test findings are compared and showed that comparable trends are seen in results with prior research.



Variation of cold air temperature with respect to change in hot end area for different conical valve angles for orifice diameter 6mm.

Ratnesh Sahu et.al. (2012) At the current time, its ecolabel nature, which meets our fundamental requirements without harming nature, is the first and main characteristic of research or development. Environmental safety has now become a key element of common businesses and individuals. This article seeks to improve the effectiveness of a system known as a vortex tube used to cool industrially and to cool processes, including spot cooling, welding cooling, plastic slitting, extrusion cooling, food chilling etc. The cooling methods frequently utilised involve gas and liquids that either damage the ozone layer or contribute to global warming in the same way as CO2. In order to get the greatest output from the C.O.P. and understanding of the vortex tube, efforts have been made to incorporate many factors. The paper details the work and construction of a vortic tube with experimental findings for a number of physical, thermal and mechanical circumstances. The report contains comprehensive information. This article provides a summary of cooling and heating impact analysis, difference in temperature and C.O.P. with various working circumstances and building characteristics. The table of data with experimental values in this article also.

Variation of C.O.P with pressure in both the condition-



Mohammad O. Hamdan et.al. (2011) This study examines the effect of different operating circumstances on the vortex pipe's thermal performance. The results of the experiments indicate that the intake and cold fraction are the most important variables influencing the vortex pipe's performance. The results of the experiments indicate that insulation has minimal effect on the vortex tube's performance. The separation of energy increases as the number of intake pins increases, according to the same inlet pressure measurements”.

2.2. Conclusion of Literature Review

“The intake pressure is the required driving unit for energy separation in a vortex tube. The temperature difference of the output streams is the higher the input pressure. The effectiveness of the vortex tube also depends on the cold fraction. Energy separation and separation of energy flow efficiencies are suited to measure the distinctive characteristics of the vortex rod for the recovery of the parameters. As the length of the vortex tube grows, but only up to the crucial length, the size of the separation does not rise further in length nevertheless. The magnitude of angular velocities diminishes as the diameter of the vortex tube rises and therefore the energy separation decreases”.

3.4. Meshing the Domain

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 neighbors, 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 CATIA to increase the output of CFD staff. Until now the preprocessor 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.

3.5. Boundary Conditions & Physical Parameters

The boundary conditions are as follows:

- Inlet stagnance limits with a total pressure of 0.5422MPa (6 Bar) and a temperature total of 298 are given.
- According to experimental measures, cold exit II and static input pressures were determined.
- The cold exit-I and the hot outlet are considered to be the pressure outlet limit. The static heat exit and cold exit-I are modified such that the cold weight fractions change. Hot outlet pressure and cold outlet pressure I are maintained above cold outlet II in order to decrease flow via hot and cold ends I outlets. (iv) zero temperature gradient at hot as well as cold temperature rises; (v) no slip and adiabatic wall conditions.

3.6. Physical Parameters of the model:

For the gathering of all experimental data, used the current CFD model. In order to investigate the effect of length on the performance of vortex tubes, all geometric features of the Bramo model are kept constant in addition to the Bramo model. By varying the model length, four more variants with different lengths have been created. This prompted research into the performance of turbot tubes with a length-to-diameter ratio of 11 to 40 and six straight nozzles. With their fixed diameters, four models with L/D ratios of 11, 21.3, 32.2, and 40.7 are examined. The nozzle is set to a diameter of 3 mm. Axial holes with aperture diameters of 6 mm and 9 mm, respectively, are cold exit I and hot exits (Opening area is partially closed by conical controlled plugs to reduce the flow through). The cold output II is set to a diameter of 3 mm (Area is 7.0685mm2). Vortex Tube features six nozzle numbers and various L/D ratios for the research.

3.7. 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"

3.8. Solver

3.8.1 Governing equations

"The 3-D stable status is calculated by applying the standard K- ε turbulence model Bramo et al..[109] to the compressible turbulent high whirling flow inside the vortex tube and may therefore mimic a turbulence effect. Therefore, the conservation of mass, energy and momentum equaled the governing relations as":

-Continuity Equation

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

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

-Momentum Equation

The second law of Newton says that the rate of fluid particle momentum change is the total of the particle forces.

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

Where P is the pressure and turbulent flow pressure (Reynolds) using the following definitions correspondingly:

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

“Where μ is the fluid dynamic viscosity and δ_{ij} is Draic delta function where, $\delta_{ij} = 0$ for $i \neq j$ and $\delta_{ij} = 1$ for $i = j$ and the turbulent shear stress is defined by”:

“Boussinesq hypothesis is a common method for modeling the Reynolds stresses and it is given as follows”:

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

“Where, μ is dynamic viscosity [$N.s.m^{-2}$], $\mu_t$is turbulent viscosity, δ_{ij} isKronecker delta and k is the kinetic energy of turbulence”.

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

The term of τ_{ij} is also represented as$-\rho u_i' u_j'$, where u represents Mass averaged velocity, [ms^{-1}] and u‛‛ represents fluctuating velocity component, [ms^{-1}].

-Energy Equation

“The energy equation is derived from the first thermodynamic law, which states that the energy shift rate of a fluid particle is equal to the particle's heat rate plus its work rate”.

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

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

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

Here $\mu$is Dynamic viscosity [$N.s.m^{-2}$] and $\mu_t$is turbulent viscosity [$N.s.m^{-2}$].

If the working fluid is regarded an ideal gas, the result of the compressibility must be mandatory:

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

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:

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

$$\frac{\partial}{\partial x_j}(\rho\varepsilon u_j) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + C_{\varepsilon 1} f_{\varepsilon 1}\tau_{ij}\frac{\partial u_i}{\partial x_j}\frac{\varepsilon}{k} - f_{\varepsilon 2} C_{\varepsilon 2}\rho\frac{\varepsilon^2}{k} + \rho E$$

"Where, (σ_ε ,$C_{\varepsilon 1}$,$C_{\varepsilon 2}$) are the model constants and ($f_{\varepsilon 1}$, $f_{\varepsilon 2}$) are the damping functions,

and E is the additional term. G is the additional turbulence term and, σ_k and σ_ε are the turbulent Prandtl numbers (Pr) for *k* and *ε*".

"For K-*ε* turbulence model, the model constants, damping functions and additional terms are provided as follows".

$$\lambda = 0.11,\ \sigma_\varepsilon = 1.3,\ \sigma_k = 1.1,\ \sigma_\varepsilon = 1.3,\ C_{\varepsilon 1} = 1.44,\ C_{\varepsilon 2} = 1.92.$$

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

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

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

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

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

By using fast digital techniques for the energy equation, the Navier-Stokes equations (compressible) were combined with regulating équations to create the tendency of flow order formation and temperature transfer in the vortex tube. The pressure and temperature data collected from the tests are given as an input for the study.

To fulfil the stated limit conditions of the computational domain CFD, the governing equations are solved repeatedly. For the general preserving of the governing equations, each mistake or residual is recorded. For this simulation, the residual is set to 1 x 10-10, which implies that when the error is equal or less than 1 x 10-10, the solution converges (or the governing equation is preserved). The initial simulation for 520 iterations (until the mass flow reaches its maximum) followed by a non-convergence of the solution. A variety of variables influence how near the finished CFD solution is to the precise solution, in conjunction with the size and size of the final residues. The remaining iterations were still high after 520 iterations (1 x 10-3), indicating that the right answer would need additional iterations. The following simulation was conducted for up to 600 iterations and revealed almost little reduction in residues.

Grid Independence Test:

Current work utilises unstructured inflation mesh (boundary layer mesh). As a parameter to verify the grid sensitivity of calculated findings, the cold air temperature was utilised. The cell count varies between 60 000 and 200 000, ranging from rough to fine. Cold temperatures have decreased and reach an optimal value of 148700 cells after an increase in the fineness of the calculation domain. Thereafter, all calculations were done with a cell domain of 148700.

Smaller cell sizes provide greater performance, but on the other hand it takes longer for the system to solve. Therefore, for various areas, different cell sizes are considered, i.e. fine size is desired at the inlet and outflow passage while moderate size is recommended as for the remainder of the pipe.

Post Processing of Results

This section will assess and investigate the numerical findings of the preliminary CFD's post-processing. After that, the data from the numerical model will be processed to interactively show the information in the form of contour tracks (temperature variables), x-y graphs, and other tools (variable values in the domain). This section looks at the temperature transmission post-processing findings.

The purpose of this numerical study is to compare the numerical results subjectively and to evaluate the acceptability of these results. These numerical results are best compared by using x-y-plots. At several measurement stations at various Z/L positions, where Z is the z-coal (tube axis) and L is the tube length, the rotational and radial pressures profiles are shown.

In the final evaluation of stalls, casements, and bridges, the FEM is used to regulate its use in the field of structural construction. The goal of dynamic structure research is to find regular frequencies, modes, and structural reactions. In the static and dynamic presentation of its structures, such as atomic vessels, the control structure, and the dynamic reaction of the reactor segment regulatory structure, the design also employs limited component technological ideas. Furthermore, for impact studies on skulls, even the biological structure employs a minimal component approach. The dam supply system's underlying geomechanics may be connected to the exhumation, sub-ground, and complex investigation of its component approach.

Component strategy statement complies with the need to evaluate existing complicated structures and structures, which typically do not include closed-form arrangements to manage balancing situations.

3.11. CFD SOFTWARE – ANSYS

ANSYS was developed by Dr. John Swanson. One of the pioneers of the FEA was the idea of a PC-simulated building, which Inc. dreamed of selling in 1970. ANSYS Inc. promotes and provides adaptable, broad design frameworks that enable companies to tackle a wide range of research issues while also expanding their existing programming and equipment interests. ANSYS Inc. is a forerunner in the industry. This is their position. It also supports a procedural approach to construction and installation, allowing customers to stay away from costly, time-consuming constructing and interrupting cycles. Client usability, information similarity, multi-stage help, and multilateral scientific combination field capabilities are all features of ANSYS examination and reproduction equipment.

3.12. Evolution of ANSYS Program

ANSYS has become a multi-use research strategic programme that has taken into account many strengths worldwide. The programme is now extremely innovative and user friendly. Each discharge has new and updated features that make the software more flexible, useful and fast. ANSYS enables experts in these areas to meet current standards and requirements.

3.13. Outline of the program

The kit provides precise, durable and integrated inspection of design. On big computers and systems, the software runs from PCs to workstations and supercomputers. ANSYS provides file consistency across all platforms and software families. The customers may load plan models supported by a PC into ANSYS by means of structural information and do a rehabilitated job. This enables the broad, adaptive design responsiveness of all ANSYS users.

3.18. METHOLODGY AND MODELLING

This part aims at optimizing the air-conditioning analysis by using the computational fluid dynamics described by ANSYS. Various stages for the procedure are described in depth. Flow simulation in internal combustion engines is a complex technique, combining fluid dynamics and turbulent movement. Either an empirically or numerically resolvable physical fluid with flow issues. The numerical simulation is more suitable for parametric study and has an enhanced impact by the application of fixation control equations in every mobile fluid domain. The area of mathematical methods has grown brilliantly in recent years, having a strong effect on evaluating and responding to complicated flux problems. The computer fluid dynamics (CFD) has developed from mathematical curiosity to an essential tool in virtually every fluid-dynamic department. CFD is seen as a connection between pure experimental fluid and natural fluid theoretical dynamics. Based on time demanding and expensive tests, researchers have till these days had to interpret complex waft issues in particular. With the advent of efficient digital computers and digital simulation methods the scope of experiments required to read complex engineering issues is substantially decreased.

3.19. Computational Fluid Dynamics

CFD provides the solution of the fluid-flow equations by modeling the distinct fluid-flowing events on computers. This makes it simpler to minimise the time and effort needed to reduce the design configuration of several engineering additives. The present paintings complete a 3-dimensional numerical simulation to evaluate, utilising publicly accessible CFD software, the impact on the cylinder waft system of a 4-hour DI diesel engine of several brass chamber geometers.

3.20. Basic Structure of A CFD Code

"It might take time and impertinence to write a CFD code to answer each new fluid glide case. As a result, industry standard CFD codes such as KIVA, AVL FIRE, STAR-CD, Fluent, ANSYS CFD and Open Foam have been developed. The GUIs (graphic user interfaces) are simple for the customer to use an optimized code with a branching of trouble fixing facilities to address fluid glide, heat transfer issues etc. In three degree terms the process of creating a fluid complicates the use of the standard CFD code".

1. "Pre-Processor
2. Solver
3. Post-Processor"

3.20.1. Pre-Processor

The initial phase of the pre-processor stage is a geometric assessment of the fluid flow issue. In order to explain geometry it may complete a proven geometry-assisted schematics (CAD). Upon creation of the geometry, an appropriate mesh may be constructed. The extent to which a number of smaller volumes or components are discernable may be described as a mesh. When a realistic mesh is determined, other known characteristics of fluid include density, viscosity and turbulent portions. The cell faces, including temperature, tension, velocities, etc, are subject to limitation circumstances, as part of a stream next to an entrance, exit or wall. In improving a solution, pre-processing is the most essential step since the practical answer in particular relies on mesh quality.

3.20.3. Post-Processor

For layout analysis using secondary variables this type of stress and flow should be established. The majority of industrial CFD programmes provide a personal post-processor to compute the secondary variables and supply range for the plots using nodal data obtained from the simulation (contour in addition to line diagrams).. These include:

- Domain geometry and grid show
- Vector plots
- Line and shaded contour plots
- 2D and 3D surface plots
- View management and colour postscript output

These apps offer animation for the presentation of dynamic findings. All programmes produce dependable alphanumerical output and exportation documents for processing outside the code in addition to visuals.

3.21. ANSYS CFD (Fluid flow) Workbench Environment

"The ANSYS CFD software (fluid flow) is integrated fully into the ANSYS Working bench environment, the foundation for the full range of simulation solutions for engineering. Its adaptable design allows users to simply install anything from generic fluid glide analyses with simple and complex interactive structures. Figure 4.1 proposes a schematic arrangement of CFD fluid waft equipment with ANSYS. Overall performance can be monitored without problems at many design variables or several alternative designs may be assessed. Packs from a number of simulation disciplines may be used in the ANSYS Workbench environment for common equipment such as geometry and meshing instruments".

Heat Transfer and Radiation: ANSYS CFXD software provides a combined heat exchanger (CHT) capability for the solution of thermal conductivities in solids, besides handling the convective electricity supply by means of a fluid float. It includes a wide range of models to capture all kinds of radiative heat, whether completely or semi-transparently, radiation or opacity between and in fluids and solids.

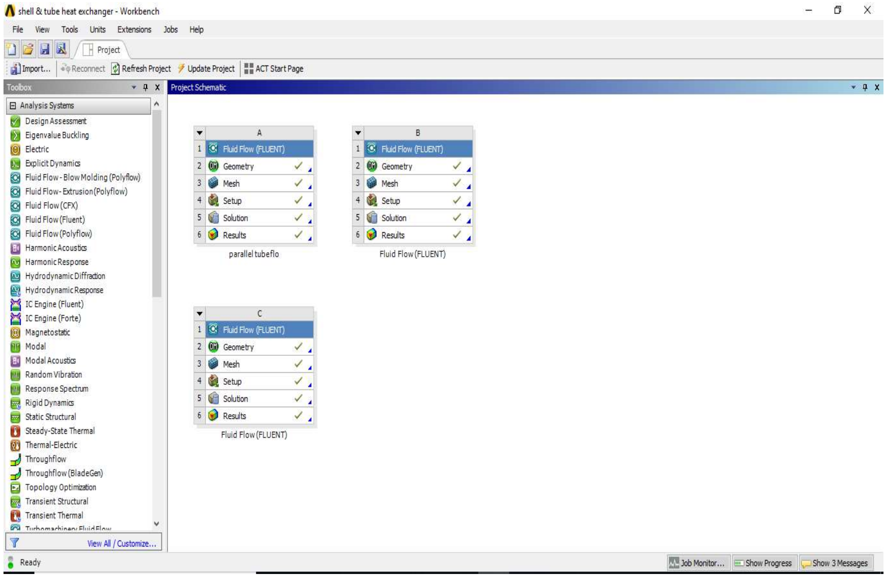


Figure 3.1: ANSYS workbench project schematic with fluid flow (CFD)

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

- Create or import a geometry
- Building a mesh for a geometry
- Set up the evaluation a good way to be dispatched to the solver
- Control and display the solver to reap a solution
- Visualize the results in a put up-processor and create a report”.

3.22. Create a Geometry

“All simulations of engineering start with geometry, which reflects whether the layout is important for a structural evaluation or the air quantities of a fluid or electromagnetic field. The engineer has either CAD geometry (a laptop-assisted layout) or creates the geometry from scratch. The development model of ANSYS is a geometric guide for an ANSYS review". Geometry was developed to generate and teach geometry for simulations using the ANSYS Design Modeler programme. Geometry comprises features that are not required in engineering simulations for simulation. Only the physics involved must be included, which is simulated by boom solver runtime in this fully designated version. Using this information for a short period may be more effective to decrease the whole duration across hours or days.

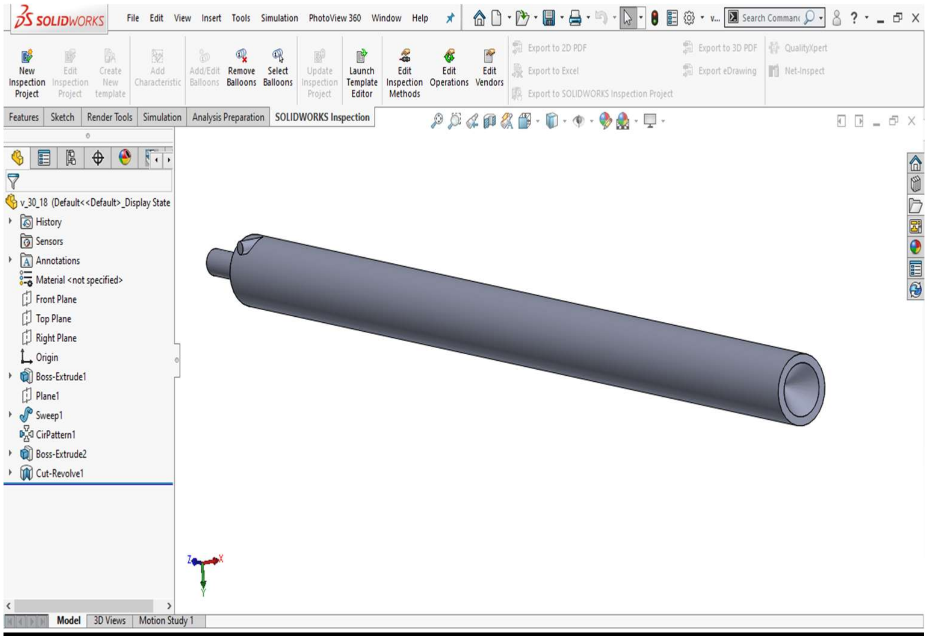


Figure 3.2: 3d model of vortex tube

3.23. Fluent(CFD)Analysis in Ansys

- "MESH RESULT
- TYPE ICEM CFD
  - Nodes 106087
  - Elements 22483
- ANALYSIS SETTING
- Analysis Type FLUENT
- Solver Target FLUENT

3.23.1. 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

3.23.2. Boundary conditions:

- Inlet: Pressure = 2,3 ,5 bar(gauge)
- Hot outlet: Pressure = 1 bar(gauge)
- Cold outlet: 0 bar(gauge)”

CHAPTER 4

CALCULATION

4.1. Introduction

This section will provide an outline of the project's experimental approach used to address the issues and gaps in the existing literature. Dimensional analysis, as well as the design of physical models and test rig setup, were given preliminary attention. The velocimetry methods of particle tracking velocimetry and particle streak velocimetry, which were developed depending on whether the experiment is conducted in the far-field or near-field of the vortex, are then given in more detail.

The testing methods for determining the impacts of approach flow geometry, free-surface profiles, critical air core diameter, velocity fields, and flow visualization are detailed. Finally, qualitative and quantitative observations on the anticipated relative uncertainty of experimental results are addressed.

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. Study of Output Temperatures (IN K)

In this study we optimized the output data of temperatures from vortex tube as per variations in L/D ration with Pressure variations from 2 bar, 3 bar and 5 bar.

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

Table 4.1: Temperatures at different pressure with 30^0angle

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	330	294	325
3 Bar	276	298	278	310	274	300
5 Bar	268	300	265	298	270	298

Table 4.2: Temperatures at different pressure with 45^0 angle

PRESSURE	VALVE ANGLE=45^0					
	L/D=10		L/D=14		L/D=18	
	COLD EXIT	HOT EXIT	COLD EXIT	HOT EXIT	COLD EXIT	HOT EXIT
2 Bar	288	325	292	330	296	328
3 Bar	276	298	278	310	274	300
5 Bar	268	300	265	298	269	298

Table 4.3: Temperatures at different pressure with 60^0 angle

PRESSURE	VALVE ANGLE=60^0					
	L/D=10		L/D=14		L/D=18	
	COLD EXIT	HOT EXIT	COLD EXIT	HOT EXIT	COLD EXIT	HOT EXIT
2 Bar	290	323	292	326	295	323
3 Bar	274	300	280	306	286	306
5 Bar	268	300	265	300	271	298

- The different value of **cold exit temperature (T8),** which we are going to use in refrigeration system to calculate.
- **Energy balance**
- **Exergy balance**
- **COP of RHVTC** system and **VCC** system
- **Exergy efficiency** of **RHVTC** system and **VCC** system.

"The various results are obtained from the analysis fluid flow in vortex tube is discussed in this chapter. The temperature at the cold end and hot end is taken as output".

4.3. Results of the calculations

As per study, we received the output results from analysis and then we arranged all results in a table for better comparison. Tables designed as per different refrigeration uses.

Energy Balance Output parameters

Table 4.4:Energy Balance Output parameters with refrigerant 1234yf

Refrigerant	**Valve Angles**		
1234yf	**L/D=10**	**L/D=14**	**L/D=18**
	30^0	**45^0**	**60^0**
Evaporator	43.96	157	43.96
Condenser	171.9	399.1	318.8
Heat Exchanger	5.652	11.1	5.652
Compressor	158.7	272.9	305.6
Rhvtc	326.8	334.8	344.9

Table 4.5: Energy Balance Output parameters with refrigerant 1234ze

Refrigerant	**Valve angle**		
1234 ze	**L/d=10**	**L/d=14**	**L/d=18**
	30^0	**45^0**	**60^0**
Evaporator	157	43.96	157
Condenser	300.6	261.5	462.1
Heat exchanger	11.1	19.47	11.1
Compressor	174.4	248.3	335.8
Rhvtc	316.7	327.8	334.8

Table 4.6:Energy Balance Output parameters with refrigerant R32

Refrigerant	**Valve angle**		
R32	**L/d=10**	**L/d=14**	**L/d=18**
	30^0	**45^0**	**60^0**
Evaporator	43.96	157	43.96
Condenser	171.9	217.4	318.8
Heat exchanger	19.47	25.32	19.47
Compressor	158.7	280.4	305.6
Rhvtc	309.7	354.6	327.8

Table 4.7:Energy Balance Output parameters with Isobutane

Refrigerant	**Valve angle**		
Isobutane	**L/d=10**	**L/d=14**	**L/d=18**
	30^0	**45^0**	**60^0**
Evaporator	366.4	366.4	366.4
Condenser	539	653.9	727.4
Heat exchanger	36.22	36.22	36.22
Compressor	203.4	318.4	391.8
Rhvtc	296.6	314.7	314.7

4.4. Exergy Balance

Exergy Balance Output parameters

Table 4.8: Exergy Balance Output parameters with refrigerant 1234yf

Refrigerant	**Valve angle**		
1234yf	**L/d=10**	**L/d=14**	**L/d=18**
	30^0	**45^0**	**60^0**
Evaporator	36.35	36.35	36.36
Condenser	140.2	222.1	274.5
Heat exchanger	14.29	14.29	14.29
Compressor	0.5863	0.5863	0.5863
Rhvtc	311.5	330.9	330.9
Throttling valve	54.23	54.23	54.23

Table 4.9: Exergy Balance Output parameters with refrigerant 1234ze

Refrigerant	**Valve angle**		
1234 ze	**L/d=10**	**L/d=14**	**L/d=18**
	30^0	**45^0**	**60^0**
Evaporator	83.58	83.58	83.58
Condenser	257.9	347.9	405.5
Heat exchanger	11.71	11.71	11.71
Compressor	0.5863	0.5863	0.5863
Rhvtc	276.2	293.8	293.8
Throttling valve	54.23	54.23	54.23

In above explained tables we used various refrigerants in study of Energy and exergy analysis using vortex tube with variations in refrigerant and different pressures. Also we changed the L/d ratios of studying effect of output parameters.

Table 4.10: Exergy Balance Output parameters with refrigerant R32

Refrigerant	**Valve angle**		
R32	**L/d=10**	**L/d=14**	**L/d=18**
	30^0	**45^0**	**60^0**
Evaporator	36.35	36.35	36.35
Condenser	140.2	222.1	274.5
Heat exchanger	10.42	10.42	10.42
Compressor	0.5863	0.5863	0.5863
Rhvtc	304.2	323.6	323.6
Throttling valve	54.23	54.23	54.23

Table 4.11: Exergy Balance Output parameters with refrigerant R32

Refrigerant	**Valve angle**		
Isobutane	**L/d=10**	**L/d=14**	**L/d=18**
	30^0	**45^0**	**60^0**
Evaporator	348.9	348.9	348.9
Condenser	475.8	580.9	648
Heat exchanger	7.833	7.833	7.833
Compressor	0.5863	0.5863	0.5863
Rhvtc	229.9	244.4	244.4
Throttling valve	54.23	54.23	54.23

4.5. COP of RHVTC System

Table 4.12: COP of RHVT system at 30^0 valve angles with different inlet pressure such as 2, 3 , 5 bar at different length of the VT

Refrigerant	**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**
1234yf	0.2476	0.2481	0.2456	0.2475	0.2485	0.248	0.2532	0.2524	0.2532
1234ze	0.288	0.289	0.2887	0.2878	0.2912	0.286	0.2932	0.2921	0.2954
R32	0.2474	0.2463	0.2465	0.2473	0.2531	0.2435	0.2534	0.2532	0.262
Isobutane	0.3618	0.3652	0.3632	0.3617	0.3656	0.3621	0.3654	0.3632	0.3668

AT 30-degree valve angle, we have seen the COP OF RHVTC system with refrigerant ISOBUTANE at 5 bars with L/D ratio 18 is higher than other refrigerant.

Table 4.13: COP of RHVT system at 45^0 valve angles

Refrigerant	**Valve angle=45^0**								
	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**
1234yf	0.2379	0.2382	0.2384	0.2380	0.2386	0.2388	0.2378	0.2377	0.2380
1234ze	0.2756	0.2758	0.2760	0.2759	0.2760	0.2763	0.2758	0.2760	0.2767
R32	0.2375	0.2377	0.2380	0.2376	0.2379	0.2383	0.2375	0.2378	0.2382
Isobutane	0.3436	0.3438	0.3441	0.3437	0.3442	0.3448	0.3438	0.3443	0.3448

AT 45-degree valve angle, we have seen the COP OF RHVTC system with refrigerant ISOBUTANE at 5 bars with L/D ratio 14 is higher than other refrigerant.

Table 4.14: COP of RHVT system at 60^0 valve angle

Refrigerant	**Valve angle=60^0**								
	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**
1234yf	0.2321	0.2324	0.2326	0.2322	0.2325	0.2329	0.2324	0.2326	0.2331
1234ze	0.2682	0.2684	0.2686	0.2683	0.2685	0.2688	0.2684	0.2687	0.2692
R32	0.2316	0.2319	0.2321	0.2318	0.2320	0.2323	0.2319	0.2321	0.2324
Isobutane	0.333	0.3331	0.3332	0.3331	0.3334	0.3336	0.3332	0.3334	0.3337

AT 60-degree valve angle, we have seen the COP OF RHVTC system with refrigerant ISOBUTANE at 5 bars with L/D ratio 18 is higher than other refrigerant.

4.6. Pressure (IN Pascal)

Table 4.15: Pressure values at cold outlet with 30^0 ,45^0,60^0Valve angle

Angle									
	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**
30^0	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
45^0	2.68e+4	3.22e+4	2.08e+4	2.5e+4	3.49e+4	4.84e+4	2.52+4	3.44e+4	4.86e+4
60^0	2.81e+4	3.76e+4	4.75e+4	2.73e+4	3.85e+4	5.2e+4	2.58e+4	3.53e+4	5.33e+4

4.7. COP Of VCC System

Table 4.16:COP of VCC system at 30^0 valve angles with different inlet pressure such as 2 3 5 bars at different length of the VT

Refrigerant	**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**
1234yf	0.2244	0.2244	0.2244	0.2244	0.2244	0.2244	0.2244	0.2244	0.2244
1234ze	0.7293	0.7293	0.7293	0.7293	0.7293	0.7293	0.7293	0.7293	0.7293
R32	0.2244	0.2244	0.2244	0.2244	0.2244	0.2244	0.2244	0.2244	0.2244
Isobutane	1.459	1.462	1.460	1.459	1.460	1.463	1.459	1.459	1.462

AT 30-degree valve angle, we have seen the COP OF VCC system with refrigerant ISOBUTANE at 5 bars with L/D ratio 14 is higher than other refrigerant

Table 4.17: Pressure values at outlet with 45^0 Valve angle

Refrigerant	**Valve angle=45^0**								
	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**
1234yf	0.1434	0.1453	0.1456	0.1424	0.1421	0.1386	0.1390	0.1406	0.1415
1234ze	0.466	0.4665	0.4668	0.4658	0.4660	0.4662	0.4663	0.4664	0.4667
R32	0.1436	0.1438	0.1439	0.1440	0.1438	0.1442	0.1445	0.1438	0.1447
Isobutane	0.9321	0.9328	0.9332	0.9322	0.9326	0.9334	0.9326	0.9332	0.9336

AT 45-degree valve angle, we have seen the COP OF VCC system with refrigerant ISOBUTANE at 5 bars with L/D ratio 18 is higher than other refrigerant

Table 4.18: Pressure values at outlet with 60^0 Valve angle

Refrigerant	**Valve angle=60^0**								
	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**
1234YF	0.1165	0.1170	0.1176	0.1168	0.1175	0.1182	0.1171	0.1175	0.1186
1234ZE	0.3787	0.3790	0.3794	0.3788	0.3792	0.3797	0.3785	0.3788	0.3795
R32	0.1165	0.1172	0.1178	0.1168	0.1178	0.1184	0.1172	0.1178	0.1188
Isobutane	0.7574	0.7578	0.7580	0.7576	0.7582	0.7586	0.7578	0.7580	0.7584

AT 60-degree valve angle, we have seen the COP OF VCC system with refrigerant ISOBUTANE at 5 bars with L/D ratio 18 is higher than other refrigerant

4.8. Exergetic efficiency of RHVTC system

Table 4.19: Exegetic efficiency of RHVTC system at 30^0 valve angles with different inlet pressure such as 2 3 5 bars at different length of the VT

Refrigerant	**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**
1234yf	0.04659	0.04662	0.04667	0.04661	0.04668	0.0467	0.04658	0.04662	0.04665
1234ze	0.03404	0.03406	0.03411	0.03405	0.03407	0.03413	0.03404	0.03409	0.03415
R32	0.01353	0.01355	0.01358	0.01354	0.01356	0.01362	0.01355	0.01358	0.01364
Isobutane	0.0167	0.01672	0.01674	0.01671	0.01674	0.01676	0.01673	0.01675	0.01678

AT 30-degree valve angle, we have seen the Exegetic efficiency of RHVTC system with refrigerant R-1234yf at 3 bars with L/D ratio 14 is higher than other refrigerant

Table 4.20: Exegetic efficiency of RHVTC system with 45^0 Valve angle

Refrigerant	**Valve angle=45^0**								
	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**
1234YF	0.05362	0.05364	0.05368	0.05363	0.05365	0.05369	0.05362	0.05366	0.05370
1234ZE	0.03863	0.03864	0.03866	0.03864	0.03867	0.03872	0.03865	0.03868	0.03872
R32	0.01583	0.01585	0.01588	0.01584	0.01588	0.01591	0.01585	0.01586	0.01594
Isobutane	0.01829	0.01831	0.01835	0.01831	0.01835	0.01838	0.01832	0.01836	0.01840

AT 45-degree valve angle, we have seen the Exegetic efficiency of RHVTC system with refrigerant R-1234yf at 5 bars with L/D ratio 18 is higher than other refrigerant

Table 4.21: Exegetic efficiency of RHVTC system with 60^0 Valve angle

Refrigerant	**Valve angle=60^0**								
	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**
1234yf	0.05804	0.05806	0.05810	0.05806	0.05809	0.05812	0.05805	0.05807	0.05810
1234ze	0.0416	0.04162	0.04165	0.04162	0.04167	0.04171	0.04163	0.04169	0.0473
R32	0.01731	0.01734	0.01736	0.01732	0.01737	0.01740	0.01733	0.01738	0.01742
Isobutane	0.01931	0.01933	0.01936	0.01933	0.01938	0.01940	0.01936	0.01939	0.01943

AT 60-degree valve angle, we have seen the Exegetic efficiency of RHVTC system with refrigerant R-1234yf at 5 bars with L/D ratio 14 is higher than other refrigerant.

4.9. Exergetic efficiency of VCC system

Table 4.22: : Exegetic efficiency of RHVTC system at 30^0 valve angles with different inlet pressure such as 2, 3 ,5 bars at different length of the VT

Refrigerant	**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**
1234YF	0.8096	0.8096	0.8096	0.8096	0.8096	0.8096	0.8096	0.8096	0.8096
1234ZE	1.84	1.84	1.84	1.84	1.84	1.84	1.84	1.84	1.84
R32	0.8096	0.8096	0.8096	0.8096	0.8096	0.8096	0.8096	0.8096	0.8096
ISOBUTANE	3.502	3.502	3.502	3.502	3.502	3.502	3.502	3.502	3.502

AT 30-degree valve angle, we have seen the Exegetic efficiency of VCC system with refrigerant ISOBUTANE at 5 bars with L/D ratio 14 is higher than other refrigerant

Table 4.23: Exegetic efficiency of VCC system with 45^0 Valve angle

Refrigerant	**Valve angle=45^0**								
	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**
1234yf	0.7846	0.7846	0.7846	0.7846	0.7846	0.7846	0.7846	0.7846	0.7846
1234ze	1.443	1.443	1.443	1.443	1.443	1.443	1.443	1.443	1.443
R32	0.7846	0.7846	0.7846	0.7846	0.7846	0.7846	0.7846	0.7846	0.7846
Isobutane	2.505	2.505	2.505	2.505	2.505	2.505	2.505	2.505	2.505

AT 45-degree valve angle, we have seen the Exegetic efficiency of VCC system with refrigerant ISOBUTANE at 5 bars with L/D ratio 14 is higher than other refrigerant

Table 4.24: Exegetic efficiency of VCC system with 60^0 Valve angle

Refrigerant	**Valve angle=60^0**								
	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**
1234yf	0.7764	0.7764	0.7764	0.7764	0.7764	0.7764	0.7764	0.7764	0.7764
1234ze	1.312	1.312	1.312	1.312	1.312	1.312	1.312	1.312	1.312
R32	0.7764	0.7764	0.7764	0.7764	0.7764	0.7764	0.7764	0.7764	0.7764
Isobutane	2.174	2.174	2.174	2.174	2.174	2.174	2.174	2.174	2.174

AT 60-degree valve angle, we have seen the Exegetic efficiency of VCC system with refrigerant ISOBUTANE at 5 bars with L/D ratio 14 is higher than other refrigerant

4.10. Plotted Graphs

In section we have compared the output results with the help of plotted graphs. As below figures shows the variations in output parameters.

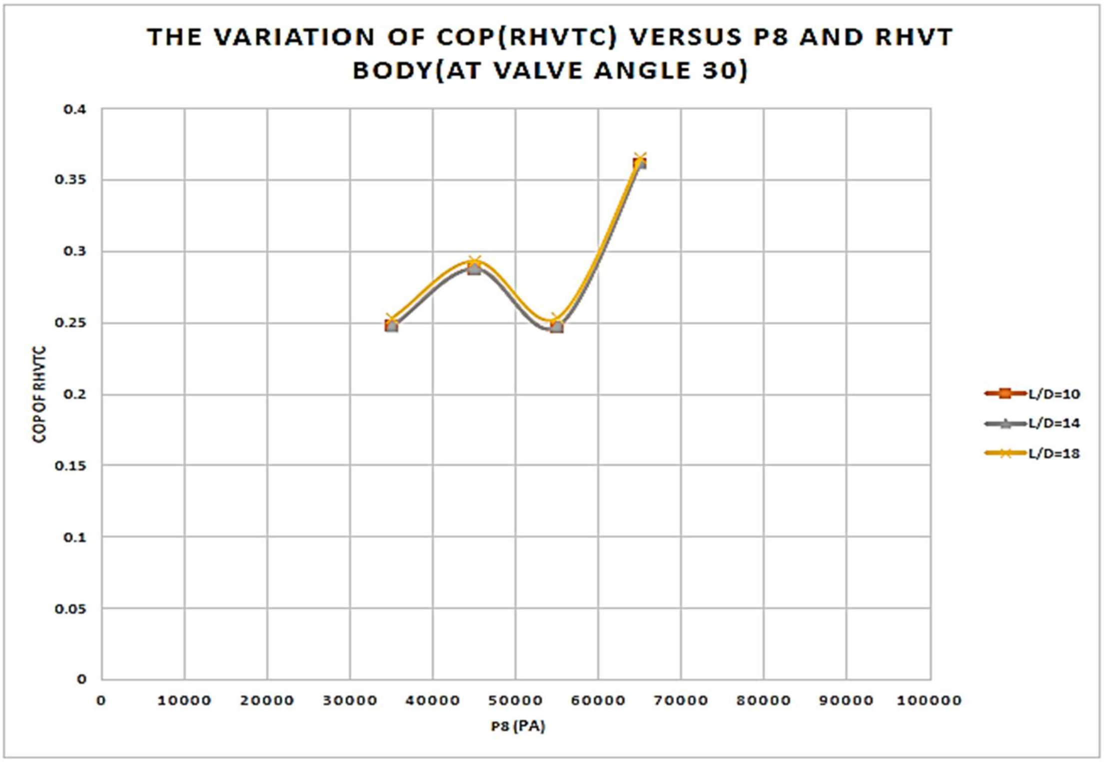


Figure 4.1: The variations of COP (RHVTC) Vs P8 and RHVT Body (at Valve Angle 30^0)

In this graph at valve angle 30^0 the COP of rhvtc is slightly higher at l/d ratio 18 as compare to other tube length.

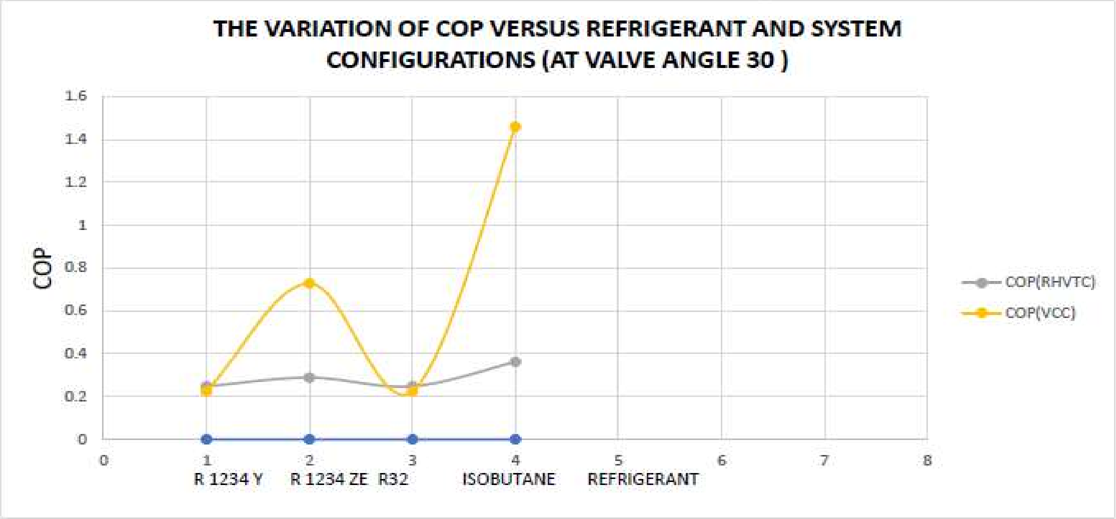


Figure 4.2: The variations of COP Vs Refrigerant and System Configurations (at Valve Angle 30^0)

this graph shows comparison between COP of rhvtc system with VCC system at valve angle 30^0 whereas for isobutane COP for both system is higher than others.

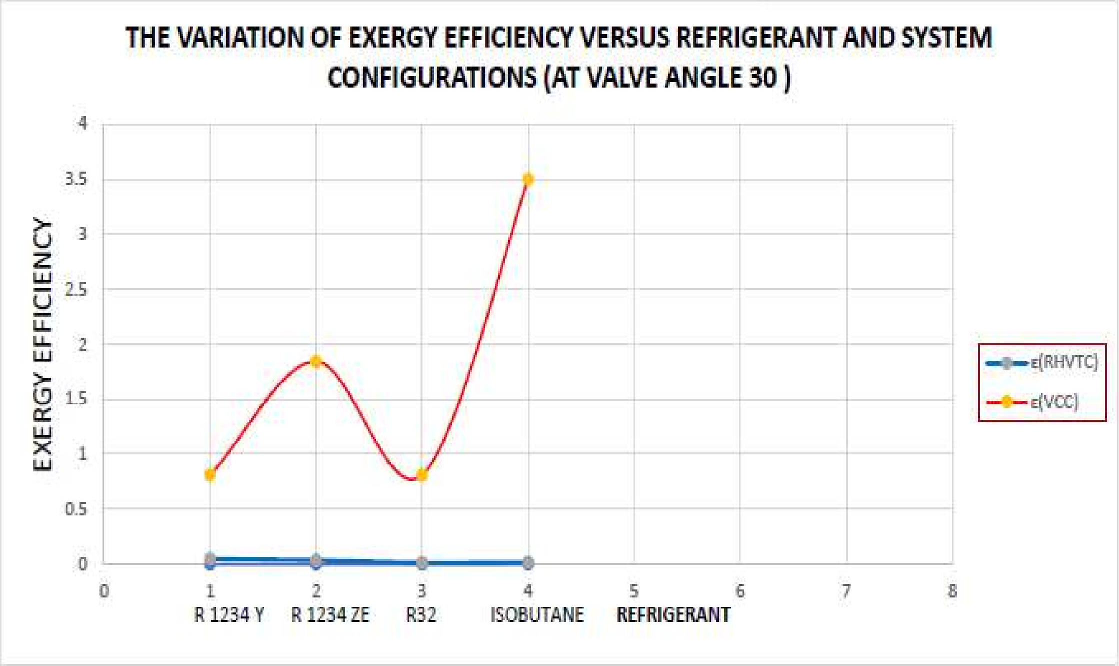


Figure 4.3: The variations of Exergy Efficiency Vs Refrigerant and System Configurations (at Valve Angle 30^0)

This graph shows comparison between exergy efficiency of rhvtc system with vcc system at valve angle 30^0whereas for isobutane exergy efficiency of rhvtc system is higher with refrigerant r-1234yf and exergy efficiency of vcc system is higher with refrigerant isobutane

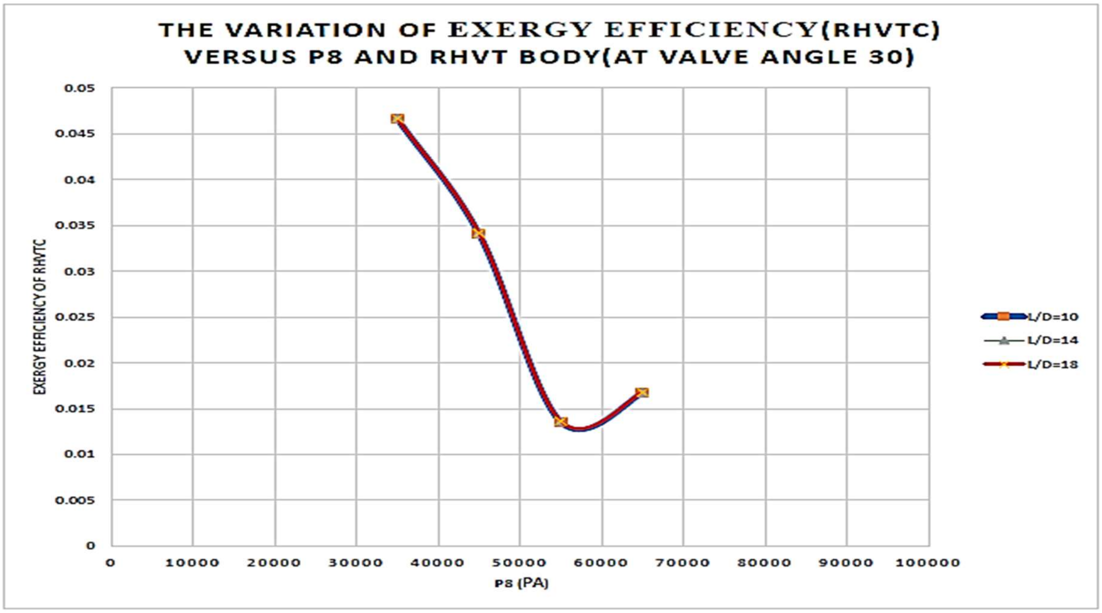


Figure 4.4: The variations of Exergy Efficiency (RHVTC) Vs P8 and RHVT Body (at Valve Angle 30^0)

the exergy efficiency is higher at l/d ratio 14 at 3 bar pressure ,30^0 valve angle.

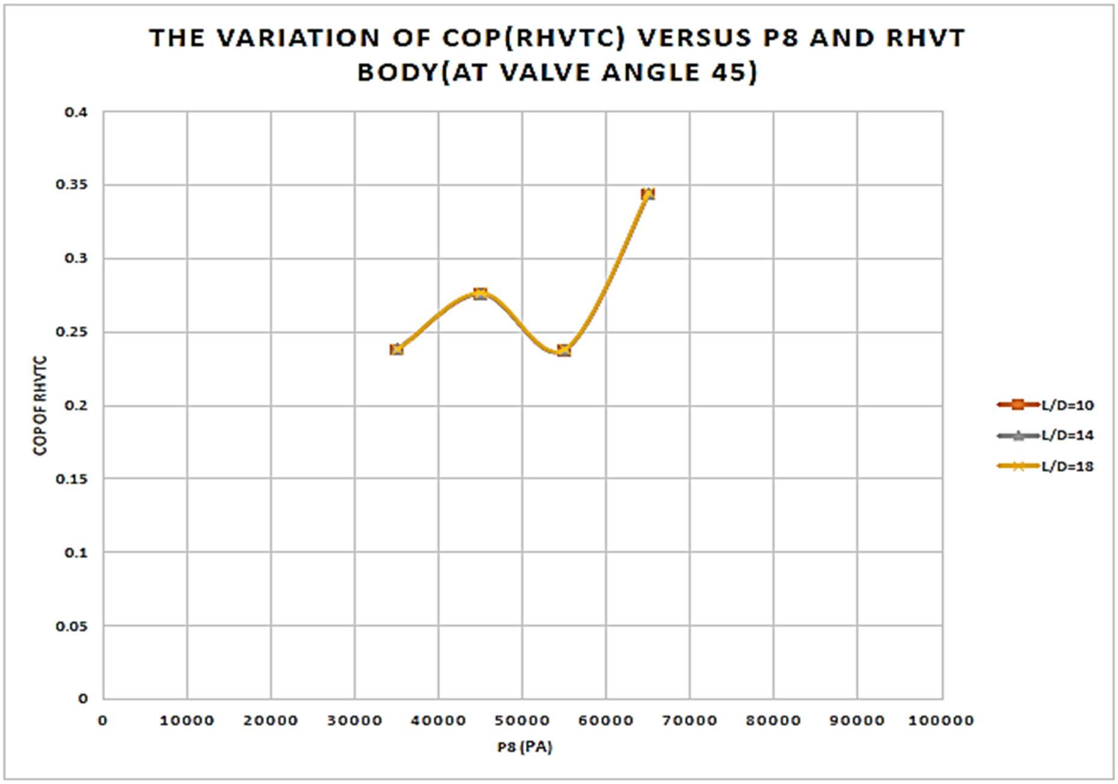


Figure 4.5: The variations of COP (RHVTC) Vs P8 and RHVT Body (at Valve Angle 45^0)

In this graph at valve angle $45^0$the cop of rhvtc is slightly higher at l/d ratio 18 as compare to other tube length.

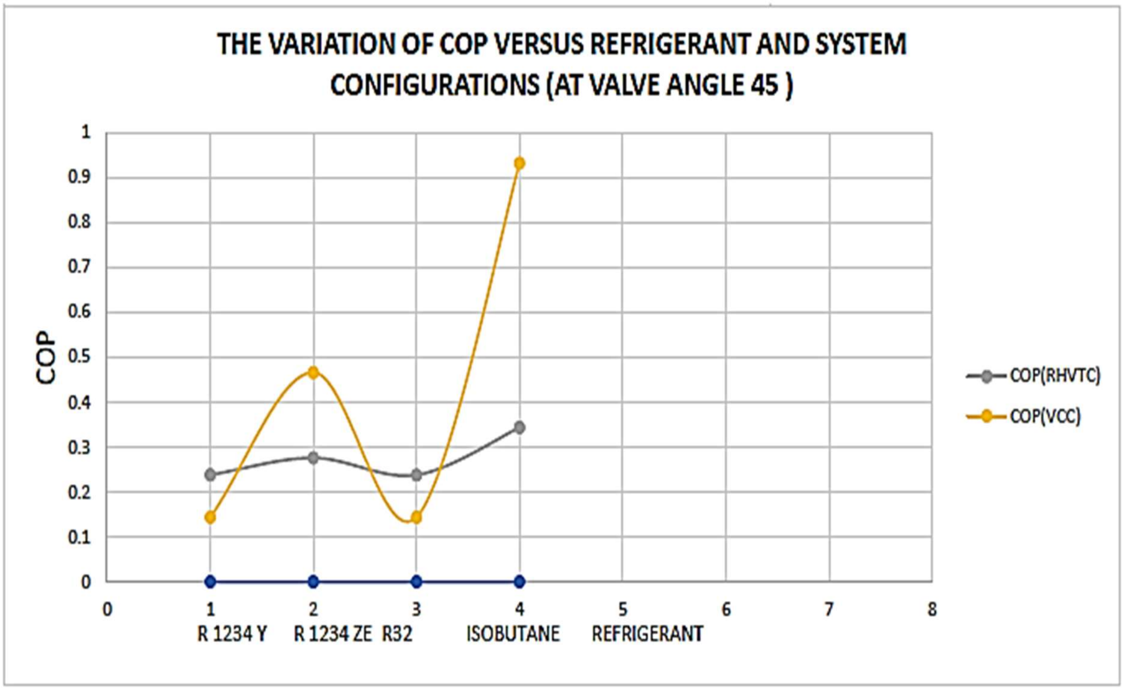


Figure 4.6: The variations of COP Vs Refrigerant and System Configurations (at Valve Angle 45^0)

This graph shows comparison between cop of rhvtc system with vcc system at valve angle 45^0 whereas for isobutane cop for both systems is higher than others.

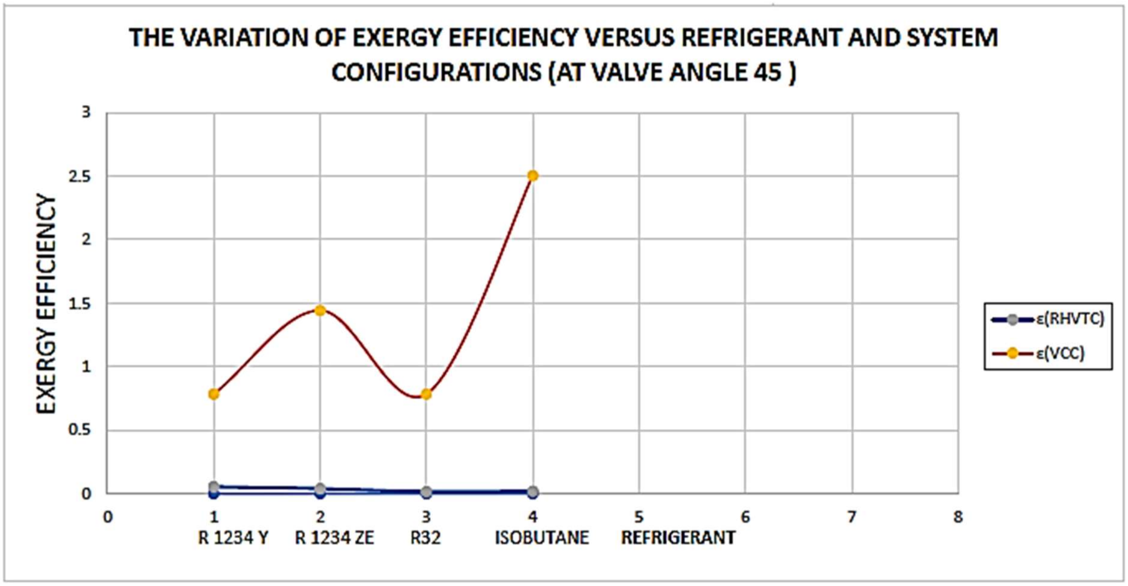


Figure 4.7: The variations of Exergy Efficiency Vs Refrigerant and System Configurations (at Valve Angle 45^0)

This graph shows comparison between exergy efficiency of rhvtc system with VCC system at valve angle 45^0 whereas for isobutane exergy efficiency of RHVTC system is higher with refrigerant r-1234yf and exergy efficiency of VCC system is higher with refrigerant isobutane.

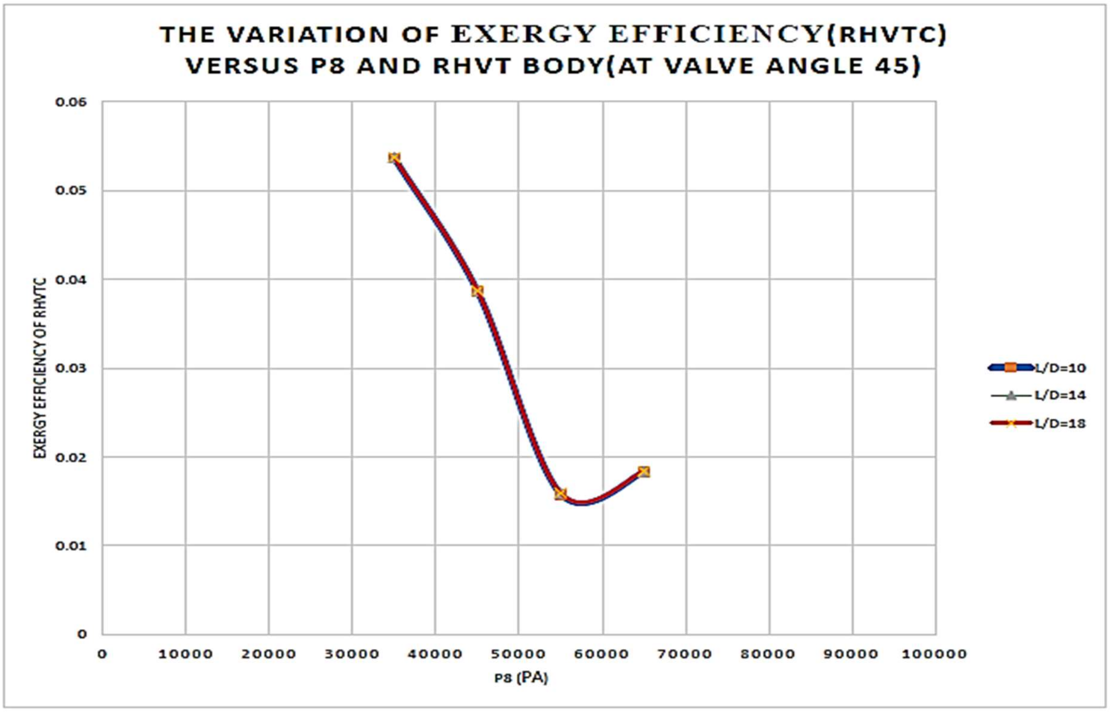


Figure 4.8: The variations of Exergy Efficiency (RHVTC) Vs P8 and RHVT Body (at Valve Angle 45^0)

The exergy efficiency is higher at l/d ratio 18 at 5 bar pressure ,45^0 valve angle.

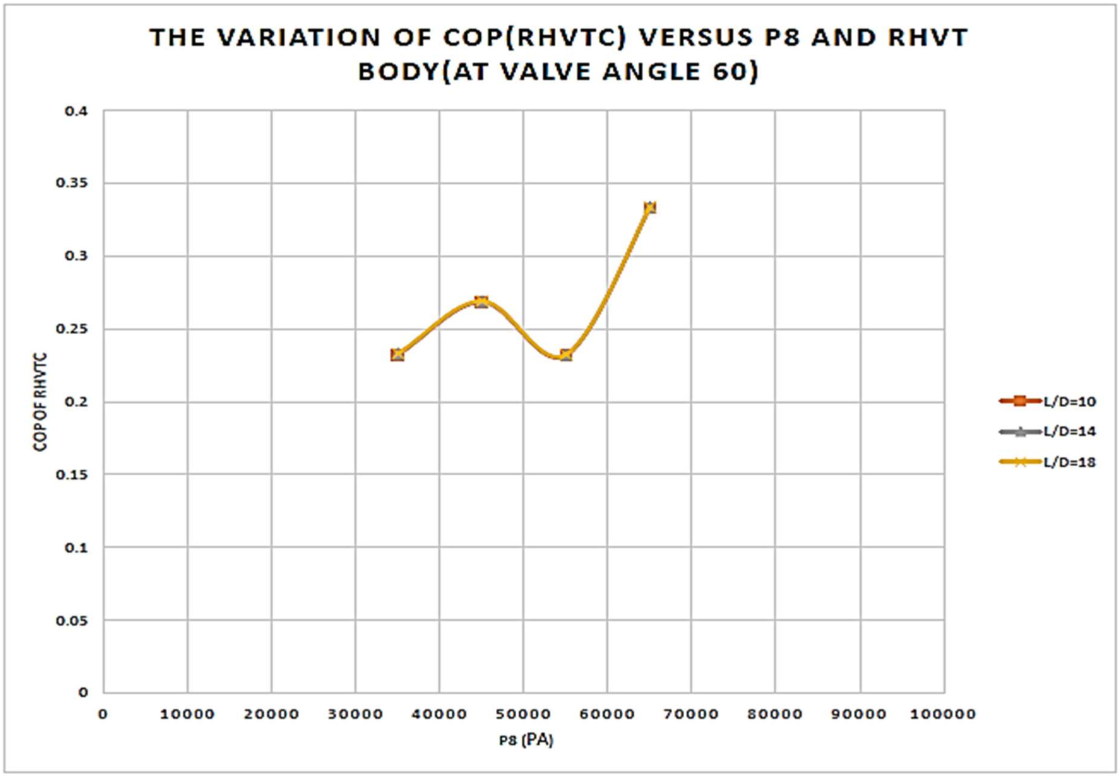
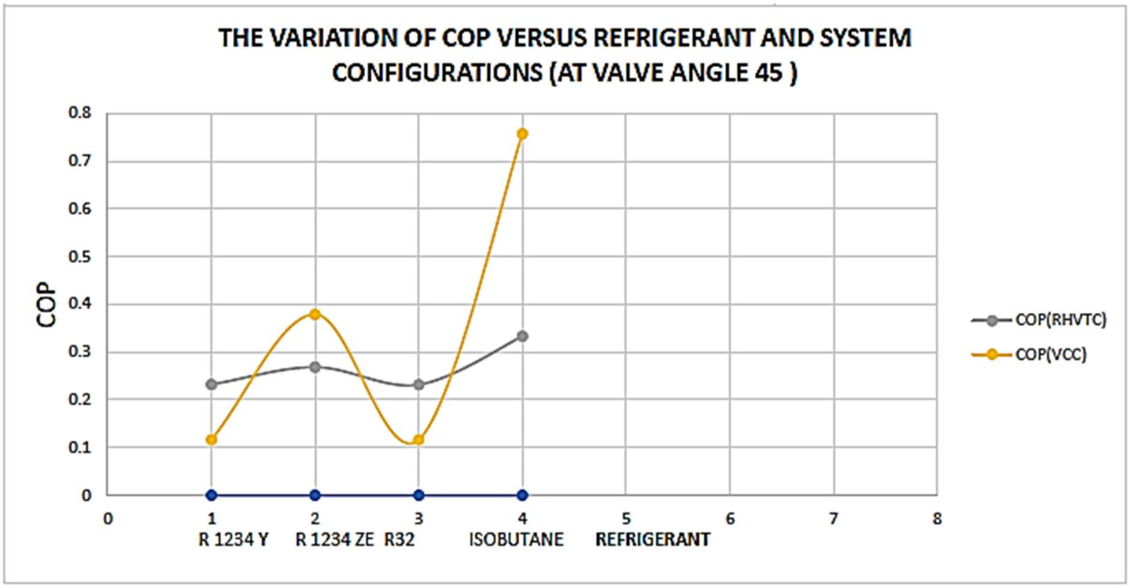


Figure 4.9: The variations of COP (RHVTC) Vs P8 and RHVT Body (at Valve Angle 60^0)

In this graph at valve angle $60^0$the COP of RHVTC is higher at l/d ratio 18 as compare to other tube length.



This graph shows comparison between COP of rhvtc system with VCC system at valve angle 60^0 whereas for isobutane COP for both systems is higher than others.

Figure 4.10: The variations of COP Vs Refrigerant and System Configurations (at Valve Angle 45^0)

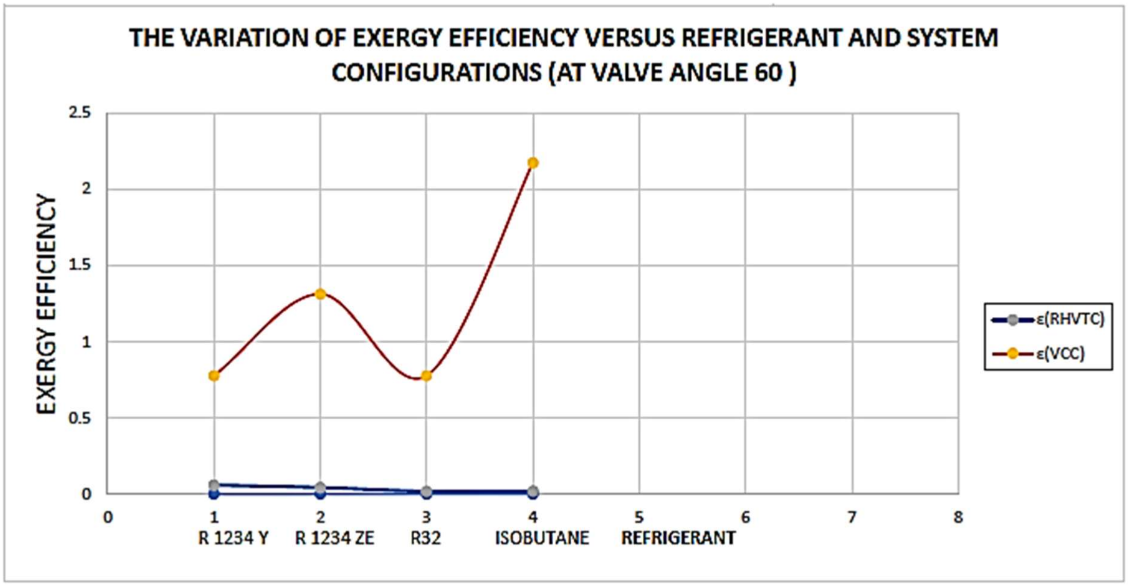


Figure 4.11: The variations of Exergy Efficiency Vs Refrigerant and System Configurations (at Valve Angle 60^0)

This graph shows comparison between exergy efficiency of RHVTC system with VCC system at valve angle 60^0 whereas for isobutane exergy efficiency of RHVTC system is higher with refrigerant r-1234yf and exergy efficiency of VCC system is higher with refrigerant isobutane

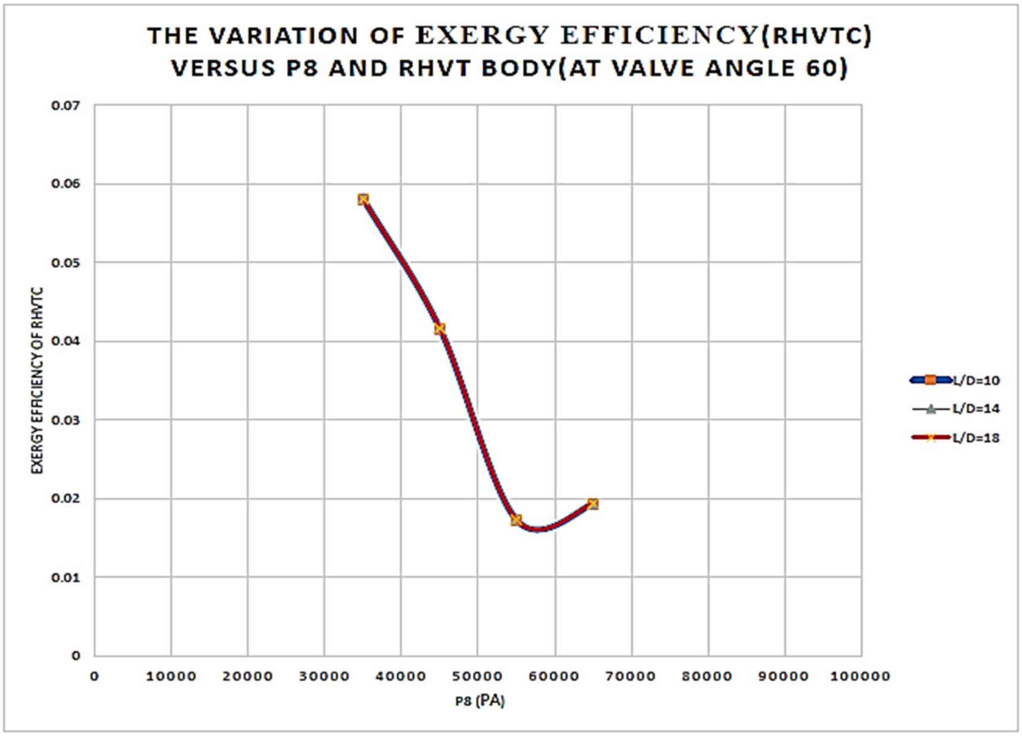


Figure 4.12: The variations of Exergy Efficiency (RHVTC) Vs P8 and RHVT Body (at Valve Angle 60^0)

The exergy efficiency is higher at l/d ratio 18 at 5 bar pressure ,60^0 valve angle.

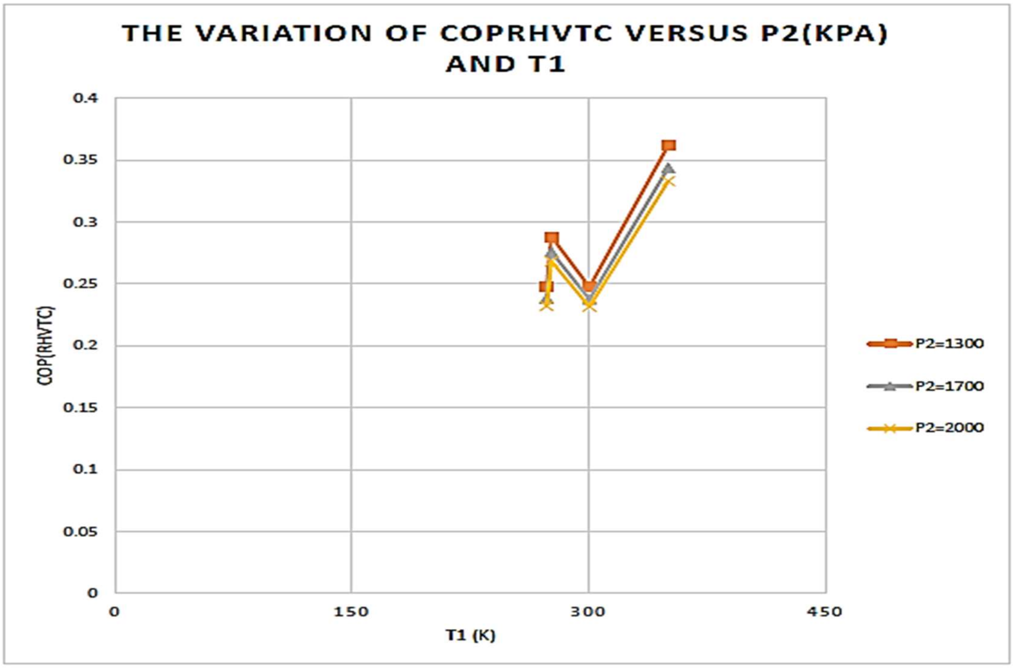


Figure 4.13: The variations of COP (RHVTC) Vs P2 (KPA) and T1

In this graph , COP of RHVTC system at three different p2=1300,1700,2000 at different t1 and we got at p2=1300 pa, t1=350.2 k the COP of RHVTC system is higher.

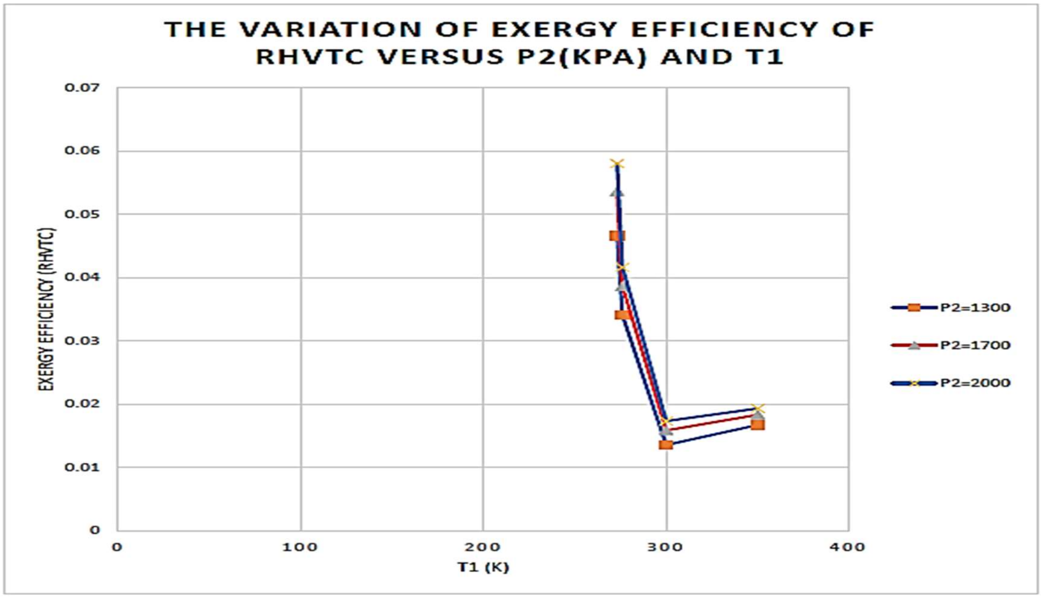


Figure 4.14: The variations of Exergy Efficiency of RHVTC Vs P2 (KPA) and T1

In this graph, exergy efficiency of RHVTC system at three different p2=1300,1700,2000 KPa at different t1 and we got at p2=2000 KPa, t1=273.2 k the exergy efficiency of rhvtc system is higher.

After analysing the COP of RHVTC system and VCC system ,exergetic efficiency of RHVTC system and VCC system

The refrigerant isobutane is suitable for the RHVTC and VCC system if we see the maximum COP and exergetic efficiency of the both system with respect to vortex tube which is our main objective to get value at different tube length ,valve angle at different inlet pressure such as 2,3 5 bars with 298k temperature of air we got maximum temperature difference at 47 degree at 30 degree valve angle ,l/d=14 at 2 bar inlet pressure.

CHAPTER 6

CONCLUSION

5.1. Conclusion

The intake pressure is the required driving unit for energy separation in a vortex tube. The temperature difference of the output streams is the higher the input pressure. The effectiveness of the vortex tube also depends on the cold fraction. Energy separation and separation of energy flow efficiencies are suited to measure the distinctive characteristics of the vortex rod for the recovery of the parameters. As the length of the vortex tube grows, but only up to the crucial length, the size of the separation does not rise further in length nevertheless. The magnitude of angular velocities diminishes as the diameter of the vortex tube rises and therefore the energy separation decreases.

“The results indicate that the CFD model used in this study accurately predicts vortex behavior in the vortex tube. The data reveals while certain errors have occurred as a consequence of the model's inaccuracy and the use of exact limit conditions, these errors may be prevented by increasing the model's accuracy. This proposed CFD model of the vortex tube was used to analyze the temperature change.

The Concluding points from project are: -

- Maximum temperature difference between hot exit and cold exit is 47 degrees Celsius at 30-degree valve angle, L/d=14 at 2 bar inlet pressure.
- We got maximum COP of RHVTC SYTEM **0.3668** at 30-degree valve angle, L/d=18 at 5 bar inlet pressure with ISOBUTANE refrigerant
- And COP OF VCC system 1.459 with ISOBUTANE refrigerant at L/D=18 AT 5 bar inlet pressure, 30-degree valve angle
- Maximum Exergy efficiency of THE RHVTC SYSTEM is 0.05810 at L/D=18, 60-degree valve angle with refrigerant R-1234YF.
- Maximum Exergy efficiency of the VCC SYSTEM is 3.502at L/D=18, 60-degree valve angle with refrigerant R-1234YF
- Maximum pressure at exits is 5.33e+04 Pascal.
- We thus succeeded in achieving fluid (air) in the vortex tube design parameters and flow analyses”.

REFERENCES

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

[2] 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.

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

[4] 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.

[5] Abu Bakar IZHAR, Arshad Hussain QURESHI and Shahab KHUSHNOOD, “Simulation of Vortex-Induced Vibrations of A Cylinder Using ANSYS CFX”, China Ocean Eng., Vol. 28, No. 4, pp. 541 – 556 © 2014 Chinese Ocean Engineering Society and Springer-Verlag Berlin Heidelberg DOI 10.1007/s13344-014-0044-1 ISSN 0890-5487.

[6] Akash S. Bidwaik and Sumit Mukund Dhavale, “To Study the Effects of Design Parameters on Vortex Tube with CFD Analysis”, International Journal of Engineering Research & Technology (IJERT) ISSN: 2278-0181 IJERTV4IS100149 www.ijert.org (This work is licensed under a Creative Commons Attribution 4.0 International License.) Vol. 4 Issue 10, October-2015.

[7] Alok Manas Dubey, Ghanshyam Das Agrawa and Suresh Kumar, “Performance evaluation and optimal configuration analysis of a transcritical carbon dioxide/propylene cascade system with vortex tube expander in high-temperature cycle”, Springer Clean Techn Environ Policy DOI 10.1007/s10098-015-0998-6.

[8] Anatoliy Khait, Alexander Noskov, Vladimir Alekhin & Aleksey Antipin, “Numerical Simulation and Visualization Of Air Flow In Ranque-Hilsch Vortex Tube”, Proceedings of the 13th International Conference on Construction Applications of Virtual Reality, 30-31 October 2013, London, UK.

[9] Anil Kumar Bodukuri,“Air Refrigeration System by Using Vortex Tube”, International Journal of Application or Innovation in Engineering & Management

(IJAIEM) Web Site: www.ijaiem.org Email: editor@ijaiem.org Volume 6, Issue 7, July 2017 ISSN 2319 – 4847.

[10] Ankita Adana and P.M. Khanwalkar, “A Review on Enhancement of Refrigeration Effect of Vortex Tube”, International Journal of Current Engineering and Technology E-ISSN 2277 – 4106, P-ISSN 2347 – 5161 ©2016 INPRESSCO®, All Rights Reserved Available at http://inpressco.com/category/ijcet Review Article 213| MIT College of Engineering, Pune, India, AMET 2016, INPRESSCO IJCET Special Issue-4 (March 2016).

[11] Arun, K. K., & Selvan, S. T. (n.d.). Optimizing the Parameter of The Vortex Tube Through CFD Analysis For Sustainable Manufacturing. International Conference on Recent Trends In Engineering And Management, 6–12.

[12] Aswalekar, U. V, Solanki, R. S., Kaul, V. S., Borkar, S. S., & Kambale, S. R. (2014). Study and Analysis of Vortex Tube. International Journal of Engineering Science Invention, 3(11), 51–55.

[13] Bazgir, A., Khosravi-Nikou, M., & Heydari, A. (2019). Numerical CFD analysis and experimental investigation of the geometric performance parameter influences on the counter-flow Ranque-Hilsch vortex tube (C-RHVT) by using optimized turbulence model. In Heat and Mass Transfer/Waerme- und Stoffuebertragung. Heat and Mass Transfer. https://doi.org/10.1007/s00231-019-02578-1.

[14] Bej, N., & Sinhamahapatra, K. P. (2014). Exergy analysis of a hot cascade type Ranque-Hilsch vortex tube using turbulence model. Energy Economics, 45(December), 13–24. https://doi.org/10.1016/j.ijrefrig.2014.05.020.

[15] Celik, A., Yilmaz, M., Kaya, M., & Karagoz, S. (2017). The experimental investigation and thermodynamic analysis of vortex tubes. Heat and Mass Transfer/Waerme- Und Stoffuebertragung, 53(2), 395–405. https://doi.org/10.1007/s00231-016-1825-2.

[16] Dhillon, A. K., & Bandyopadhyay, S. S. (2015). CFD analysis of straight and flared vortex tube. IOP Conference Series: Materials Science and Engineering, 101(1). https://doi.org/10.1088/1757-899X/101/1/012067.

[17] Di Domenico, N., Groth, C., Wade, A., Berg, T., & Biancolini, M. E. (2018). Fluid structure interaction analysis: Vortex shedding induced vibrations. Procedia Structural Integrity, 8, 422–432. https://doi.org/10.1016/j.prostr.2017.12.042.

[18] Dubey, A. M., Das Agrawal, G., & Kumar, S. (2016). Performance evaluation and optimal configuration analysis of a transcritical carbon dioxide/propylene cascade

system with vortex tube expander in high-temperature cycle. Clean Technologies and Environmental Policy, 18(1), 105–122. https://doi.org/10.1007/s10098-015-0998-6.

[19] Hamdan, M. O., Alargha, H. M., Hilal-Alnaqbi, A., & Mathew, B. (2017). 3D Numerical Investigating of Flow Field and Energy Separation in Counter-flow Vortex Tube. Proceedings of the 2nd World Congress on Momentum, Heat and Mass Transfer, 1–8. https://doi.org/10.11159/enfht17.104.

[20] Hamdan, M. O., Alawar, A., Elnajjar, E., & Siddique, W. (2011). Experimental analysis on vortex tube energy separation performance. Heat and Mass Transfer/Waerme- Und Stoffuebertragung, 47(12), 1637–1642. https://doi.org/10.1007/s00231-011-0824-6.

[21] Hitesh R. Thakare, Aniket Monde and A. D. Parekh, "3D CFD Analysis of Vortex Tube", International Journal of Modern Trends in Engineering and Research www.ijmter.com e-ISSN No.:2349-9745, Date: 2-4 July, 2015.

[22] Izhar, A. B., Qureshi, A. H., & Khushnood, S. (2014). Simulation of vortex-induced vibrations of a cylinder using ANSYS CFX. China Ocean Engineering, 28(4), 541–556. https://doi.org/10.1007/s13344-014-0044-1.

[23] 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.

[24] K.K.Arun and S.Tamil selvan, "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.

[25] 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.

[26] 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.

[27] 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.

[28] 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.

[29] 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.

[30] 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.

[31] 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.

[32] 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.

[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.

[34] Mr. Shadab Shaikh Bismillah, Prof. K. M. Mahajan and Prof. V. H. Patil, "A Review on Performance Improvement of Vortex Tube", SSN(Online)-2454-4159 Volume 4, Issue 1, January 2018 Copyright to IJARSMT www.ijarsmt.com.

[35] Mr. H R Ghan, Mr. A A Hashmi and Dr. M M Dhobe, "Analysis: Design Optimization of Vortex Tube", International Journal of Advance Research, Ideas and Innovations in Technology. © 2017, IJARIIT All Rights Reserved Page | 28 ISSN:

2454-132X Impact factor: 4.295 (Volume3, Issue2) Available online at: www.ijariit.com.

[36] Nayak, A., Satapathy, P. K., Sahoo, S. S., & Mahapatra, I. (2019). Fluid flow and performance analysis of vortex tube: a computational approach. Energy Sources, Part A: Recovery, Utilization and Environmental Effects, 0(0), 1–15. https://doi.org/10.1080/15567036.2019.1624880.

[37] Nilotpala Bej and K.P. Sinhamahapatra, "Exergy analysis of a hot cascade type RanqueHilsch vortex tube using turbulence model", Corresponding author. Tel.: þ91 9933684740. E-mail addresses: nilotpala2002@gmail.com, nilotpala2002@yahoo.co.in (N. Bej). www.iifiir.org Available online at www.sciencedirect.com ScienceDirect journal homepage: www.elsevier.com/locate/ijrefrig international journal of refrigeration 45 (2014) 13 e2 4 http://dx.doi.org/10.1016/j.ijrefrig.2014.05.020 0140-7007/© 2014 Elsevier Ltd and IIR. All rights reserved.

[38] Pavan R. Bangare, Abhijit D. Shinde, Mayur S. Shirsath, Pravin A. Taru and Prof. G. B. Salunke, "Vortex Tube", IARJSET ISSN (Online) 2393-8021 ISSN (Print) 2394-1588 International Advanced Research Journal in Science, Engineering and Technology ISO 3297:2007 Certified Vol. 5, Issue 3, March 2018 Copyright to IARJSET DOI 10.17148/IARJSET.2018.5317 102.

[39] Prof. Uday V. Aswalekar, Ritesh S. Solanki, Vedang S. Kaul , Shrutika S. Borkar and Shankar R. Kambale, "Study and Analysis of Vortex Tube", International Journal of Engineering Science Invention ISSN (Online): 2319 – 6734, ISSN (Print): 2319 – 6726 www.ijesi.org ||Volume 3 Issue 11 || November 2014 || PP.51-55.

[40] Ratnesh Sahu, Rohit Bhadoria and Deepak Patel, "Performance Analysis of a Vortex Tube by using Compressed Air", International Journal of Scientific & Engineering Research Volume 3, Issue 9, September-2012 1 ISSN 2229-5518 IJSER © 2012 http://www.ijser.org.

[41] Ramesh ganugapenta, G.R Selokar and P VenkatRathnam, "Design and Performance Evaluation of a Vortex Tube Form by a Delrin Material", International Journal of Engineering Science Invention (IJESI) ISSN (Online): 2319 – 6734, ISSN (Print): 2319 – 6726 www.ijesi.org ||Volume 7 Issue 8 Ver II || Aug 2018 || PP 33-39 www.ijesi.org 33 | Page.

[42] R C Venkatesh, S Vishal and N Arun kumar, "A Review on Vortex Tube Performance Based on Various Geometric Parameters", International Journal of

Latest Engineering and Management Research (IJLEMR) ISSN: 2455-4847 www.ijlemr.com || Volume 02 - Issue 09 || September 2017 || PP. 01-08.

[43] R. Dhineshkumar and D. Ashokumar, “Experimental Investigation for Optimizing Parameter of Vortex Tube for Sustainable Heating System”, International Journal of Engineering Research & Technology (IJERT) ISSN: 2278-0181 Published by, www.ijert.org ICITMSEE - 2018 Conference Proceedings.

[44] Sagar N. Jadhav, Vishal D. Wagh, Mahesh N. Patil, Balaji V. Kawale and P.P. Patunkar, “An Experimental Modeling And Investigations Of Vortex Tube Using UPVC Material”, International Journal Of Technology Enhancements And Emerging Engineering Research, VOL 3, ISSUE 04 98 ISSN 2347-4289 Copyright © 2015 IJTEEE.

[45] Sankar Ram T. and Anish Raj K. “An Experimental Performance Study of Vortex Tube Refrigeration System”, ISSN: 2321-9939 2013, IJEDR1302004 International Journal Of Engineering Development And Research , IJEDR (All right reserved by www.ijedr.org).

[46] Sarath Sasi and Sreejith M, “Experimental Investigation of Vortex Tube Refrigeration”, International Journal of Emerging Engineering Research and Technology Volume 2, Issue 6, September 2014, PP 176-186 ISSN 2349-4395 (Print) & ISSN 2349-4409 (Online).

[47] Sayali Darekar, Makarand Date, Shubham Chaudhari, Mangesh Dudhare and V.S. Bagade, “Review Paper On Experimental Analysis Of Vortex Tube”, International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056 Volume: 05 Issue: 01 | Jan-2018 www.irjet.net p-ISSN: 2395-0072 © 2018, IRJET | Impact Factor value: 6.171 | ISO 9001:2008 Certified Journal.

[48] Sawaibahadure, S. (2019). A Study of Thermal Performance Testing for Annular Vortex Tube with reviews . JETIR, 6(6), 227–230.

[49] Shrikrushna Nagane, Sagar Vanage, Anandsinh Anpat and Sarang Hole , “A Review on vortex tube refrigeration system”, IJISET - International Journal of Innovative Science, Engineering & Technology, Vol. 2 Issue 9, September 2015. www.ijiset.com ISSN 2348 – 7968.

[50] S.Rejin and H.Thilakan, “Experimental Analysis on Vortex Tube Refrigerator Using Different Conical Valve Angles”, International Journal of Engineering Research and Development e-ISSN: 2278-067X, p-ISSN: 2278-800X, www.ijerd.com Volume 3, Issue 4 (August 2012), PP. 33-39 33.

[51] Surjosatyo, A., Respati, A., Dafiqurrohman, H., & Muammar. (2017). Analysis of the Influence of Vortexbinder Dimension on Cyclone Separator Performance in Biomass Gasification System. Procedia Engineering, 170, 154–161. https://doi.org/10.1016/j.proeng.2017.03.036.

[52] Waraporn Rattanongphisata and Krairin Thungthong, “Improvement vortex cooling capacity by reducing hot tube surface temperature: Experiment”, Available online at www.sciencedirect.com ScienceDirect 1876-6102 © 2014 Elsevier Ltd. This is an open access article under the CC BY-NC-ND license (http://creativecommons.org/licenses/by-nc-nd/3.0/). Selection and peer-review under responsibility of the Organizing Committee of 2013 AEDCEE doi: 10.1016/j.egypro.2014.07.048 2013 International Conference on Alternative Energy in Developing Countries and Emerging Economies .

[53] Xiaojie Zhai, “Research on the Application of Vortex Tube Type of Cooling Jacket in Coal Mine”, Cite as: AIP Conference Proceedings 1864, 020220 (2017); https://doi.org/10.1063/1.4993037 Published Online: 03 August 2017.

[54] Zahari Taha, H.A. Salaam, T.M.Y.S. Tuan Ya, S.Y. Phoon, C.F. Tan and M.A. Akiah, “Vortex Tube Air Cooling: The Effect On Surface Roughness And Power Consumption In Dry Turning”, International Journal of Automotive and Mechanical Engineering (IJAME) ISSN: 2229-8649 (Print); ISSN: 2180-1606 (Online); Volume 8, pp. 1477-1586, July-December 2013 ©Universiti Malaysia Pahang DOI: http://dx.doi.org/10.15282/ijame.8.2013.34.0122.