A Review On Exprimental & Numerical Investigation Of Ranque Hilsch Vortextube

Rahul B Patel[a] , Dr. V. N. Bartaria[b] , Tausif M Shaikh[c]

Abstract-

The vortex tube or Ranque–Hilsch vortex tube is a simple device used in industry for generation of cold and hot air streams from a single compressed air supply. This simple device is very efficient in separation of air streams of different temperatures and has been the focus of investigation since the tube's discovery. Different explanations for the phenomenon of the energy separation have been proposed, however there has not been a consensus in the hypothesis. The purpose of this paper is to present a critical review of current explanations on the working concept of a vortex tube.

Although many experimental and numerical studies on the vortex tubes have been made, the physical behaviour of the flow is not fully understood due to its complexity and the lack of consistency in the experimental findings. Furthermore, several different hypotheses based on experimental, analytical, and numerical studies have been put forward to describe the thermal separation phenomenon. Hypotheses of pressure, viscosity, turbulence & temperature separation phenomenon using CFD analysis of vortex tube are discussed in the paper, and presumably, future research will benefit from this discussion.[1]

.

Index Terms—RHVT, Thermal Separation, CFD

I. INTRODUCTION

Ranque discovered the effect of vortex energy distribution and patented the first vortex tube (RHVT) in 1934. In 1946, Hilsch improved the RHVT design and its underlying principles that still remain valid today (Khodorkov et al., 2003). Aljuwayhel et al. (2005) define the vortex tube as a simple device with no moving parts that is capable of separating a high-pressure flow into two lower pressure flows of different temperatures. The device consists of a simple circular tube, one or more tangential nozzles, and a throttle valve (see Fig. 1, Cockerill, 1995).

Working principle of the counter flow RHVT can be defined as follows. Compressible fluid, which is tangentially introduced into the vortex tube from nozzles, starts to make a circular movement inside the vortex tube at high speeds, because of the cylindrical structure of the tube, depending on its inlet pressure and speed. Pressure difference occurs between tube wall and tube center because of the friction of the fluid circling at high speeds. Speed of the fluid near the tube wall is lower than the speed at the tube center, because of the effects of wall friction. As a result, fluid in the center region transfers energy to the fluid at the tube wall, depending on the geometric structure of the vortex tube. The cooled fluid leaves the vortex tube from the cold output side, by moving towards an opposite direction, compared to the main flow direction, after a stagnation point. Whereas, the heated fluid leaves the tube in the main flow direction from the other end (Dincer et al., in press). By injecting compressed air at room temperature circumferentially into a tube at high velocity, a vortex tube can produce cold air down to 223 K and hot air up to 400 K (Crocker et al., 2003).

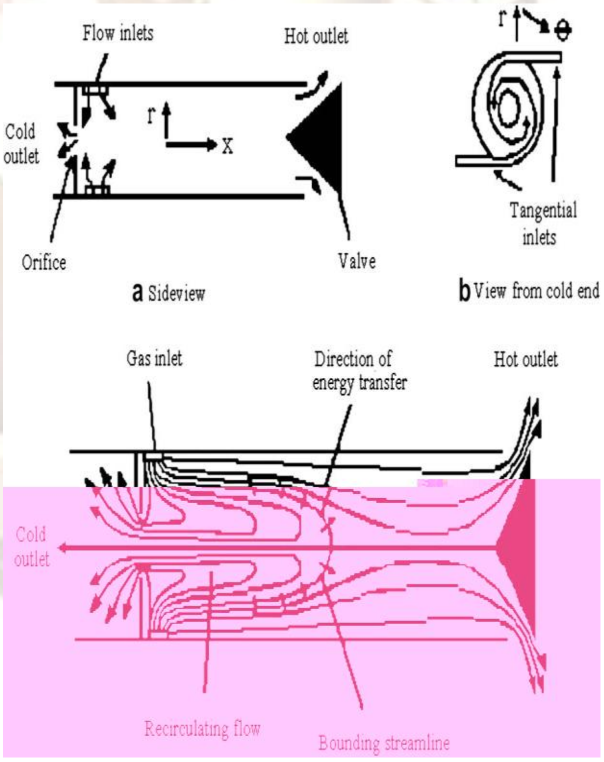

Fig. 1 – Schematic diagram of the counter flow RHVT (Cockerill, 1995).

Schematic diagram of the parallel flow RHVT is shown in Fig. 2. Both hot and cold flows in RHVTs leave the vortex tube in the same direction. It is not possible for cold flow to turn back after a stagnation point. In order to separate the flow in the center of the tube from the flow at the wall, an apparatus which has a hole in the center is used. The temperature of hot and cold flows can be changed by back and forth movement of this apparatus. In parallel flow RHVTs, hot and cold flows mix with each other. This mixing affects the temperatures of fluids negatively and causes their efficiencies to be low. For this reason, parallel flow RHVTs are generally not preferred (Dincer, 2005; Dincer et al., 2005)

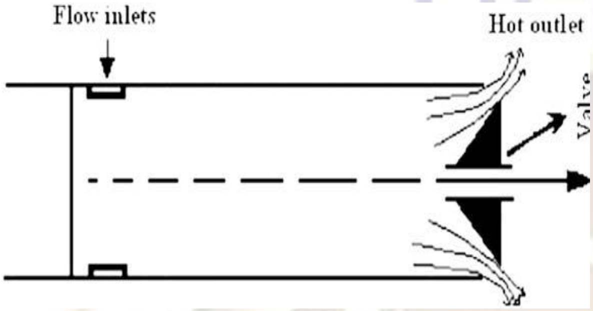

Fig. 2 – Schematic diagram of the parallel flow RHVT (Dincer et al., 2005).

In the conventional vortex tube, compressed gas enters directly a circular flow passage with equal section area from a straight pipe, which can cause a sudden change of flow direction at the joint between the straight pipe and the circular flow passage. The sudden change would lead to the generation of eddies, which cause energy loss. The sudden change of flow velocity and the eddy generated by this change can also cause the energy loss. It is important to have a high peripheral velocity in the portion of the tube immediately after the nozzle; the curve of the nozzle affects the performance of vortex tubes (Wu et al., 2007).

RHVTs are used, among others, for cooling, heating, drying and snow production. Their cooling capability is used, e.g., in dehumidifying gas samples, chilling of environmental chambers, cooling of food, welding, and air climate control processes. Although, they are not very efficient as a cooling device, they can be very useful in certain applications as they are small, simple to make and repair, and require no electrical or chemical power source. Nowadays, RHVTs are produced by different commercial companies with a wide range of applications (Dincer et al., 2008). [2]

II. INTRODUCTION TO CFD MODELLING

CFD is concerned with numerical solution of differential equations governing transport of mass, momentum and energy in moving fluid. Using CFD, one can build a computational model on which physics can be applied for getting the results. The CFD software gives one the power to model things, mesh them, give proper boundary conditions and simulate them with real world condition to obtain results. Using CFD a model can be developed which can breed to give results such that the model could be developed into an object which could be of some use in our life.

Modeling is the mathematical physics problem formulation in terms of a continuous initial boundary value problem (IBVP)
IBVP is in the form of Partial Differential Equations (PDEs) with appropriate boundary conditions and initial conditions.
Modeling includes:
1. Geometry and domain
2. Coordinates
3. Governing equations
4. Flow conditions
5. Initial and boundary conditions
6. Selection of models for different applications
Solve the Navier-Stokes equations (3D in Cartesian coordinates) .

Computational fluid dynamics techniques have revolutionized engineering design in several important areas, notably in analysis of fluid flow technology. CFD can also be used as a minimal adequate tool for design of engineering components. A careful scanning of various numerical investigations on the mechanism of thermal separation in vortex tubes indicate that barring a few ,no serious attempts have been made to use CFD techniques to simulate the flow patterns of vortex tubes.

A detailed analysis of various parameters of the vortex tube has been carried out through CFD techniques to simulate the phenomenon of flow pattern, thermal separation, pressure gradient etc. so that they are comparable with the experimental results.

The vortex tube was first observed by Ranque. Later, Hilsch did some experimental and theoretical studies to increase the efficiency of the vortex tube. Fulton explained the energy separation. He proposed that the inner layer heat the outer layer meanwhile expanding and growing cold. Reynolds did the numerical analysis of vortex tube. Lewellen arrived at the solution by combining the three Navier-stokes equations for an incompressible fluid in a strong rotating axisymmetric flow with a radial sink flow. Linderstorm-lang examined the velocity and thermal fields in the tube. He calculated the axial and radial gradients of the tangential velocity profile from prescribed secondary flow functions on the basis of a zero-order approximation to the momentum equations developed by Lewellen for an incompressible flow.

Schlenz investigated numerically in a uni-flow vortex tube, the flow field and the process of energy separation. Calculations were carried out assuming a 2D axisymmetric compressible flow and using the Galerkin's approach with a zero-equation turbulence model to solve the mass, momentum, and energy conservation equations to calculate the flow and thermal fields. Amitani used the mass, momentum and energy conservation equation in a 2D model of a counter flow vortex tube with a short length, with an assumption of a helical motion in an axial direction for an invicid compressible perfect fluid. Flohlingsdorf and Unger used the CFX system with the k-ϵ model to study the velocity profile and energy separation in a vortex tube. Aljuwahel reported that RNG k-ϵ is better than k-ϵ when he studied the energy separation and the flow phenomenon in a counter flow vortex tube using CFD in Fluent. T. Farouk and B. Farouk introduced the large eddy simulation (LES) technique to predict the flow fields and the associated temperature separation within a counter-flow vortex tube for several cold mass fractions. However, most of the computations found in the literature used simple or the first-order turbulence models which are considered unsuitable for complex, compressible vortex tube flows. The present work presents a two-dimensional numerical investigation of flow and temperature separation behaviours inside a flow vortex tube.[3]

III. EXPRIMENTAL & NUMRICAL ANALYSIS

From 1934, almost 200-215 articles are published on Ranque Hilsch vortex tube. Not all the articles are directly related to our work, especially, those articles which were focused on computational work. Many articles addressed experimental findings and remaining discussed various theories explaining energy separation phenomenon. In this subsection, we are going to discuss only those articles (experimental and/or theoretical work) which are directly related to current work.[4]

Upendra Behera et al, [5] were carried out CFD analysis and experimental investigations towards optimizing the parameters of Ranque–Hilsch vortex tube. Computational fluid dynamics (CFD) and experimental studies are conducted towards the optimization of the Ranque–Hilsch vortex tubes. Different types of nozzle profiles and number of nozzles are evaluated by CFD analysis. The swirl velocity, axial velocity and radial velocity components as well as the flow patterns including secondary circulation flow have been evaluated. The optimum cold end diameter (dc) and the length to diameter (L/D) ratios and optimum parameters for obtaining the maximum hot gas temperature and minimum cold gas temperature are obtained through CFD analysis and validated through experiments. The coefficient of performance (COP) of the vortex tube as a heat engine and as a refrigerator has been calculated.

Ahmet Murat Pinar et al, [6] were carried out Optimization of counter flow Ranque–Hilsch vortex tube performance using Taguchi method. This study discusses the application of Taguchi method in assessing maximum temperature gradient for the Ranque–Hilsch counter flow vortex tube performance. The experiments were planned based on Taguchi’s L27 orthogonal array with each trial performed under different conditions of inlet pressure, nozzle number and fluid type. Signal-to-noise ratio (S/N) analysis, analysis of variance (ANOVA) and regression analysis were carried out in order to determine the effects of process parameters and optimal factor settings. Finally, confirmation tests verified that Taguchi method achieved optimization of counter flow Ranque–Hilsch vortex tube performance with sufficient accuracy.

Tanvir Farouk et al, [7] carried out Simulation of gas species and temperature separation in the counter-flow Ranque–Hilsch vortex tube using the large eddy simulation technique.A computational fluid dynamic model is used to predict the species and temperature separation within a counter flow Ranque–Hilsch vortex tube. The large eddy simulation (LES) technique was employed for predicting the gas flow and temperature fields and the species mass fractions (nitrogen and helium) in the vortex tube. A vortex tube with a circumferential inlet stream of nitrogen–helium mixture and an axial (cold) outlet stream and a circumferential (hot) outlet stream was considered. The temporal evolutions of the axial, radial and azimuthal components of the velocity along with the temperature, pressure and mass density and species concentration fields within the vortex tube are simulated. Even though a large temperature separation was observed, only a very minimal gas separation occurred due to diffusion effects. Correlations between the fluctuating components of velocity, temperature and species mass fraction were calculated to understand the separation mechanism. The inner core flow was found to have large values of eddy heat flux and Reynolds’s stresses. Simulations were carried out for varying amounts of cold outlet mass flow rates. Performance curves (temperature separation/gas separation versus cold outlet mass fraction) were obtained for a specific vortex tube with a given inlet mass flow rate.

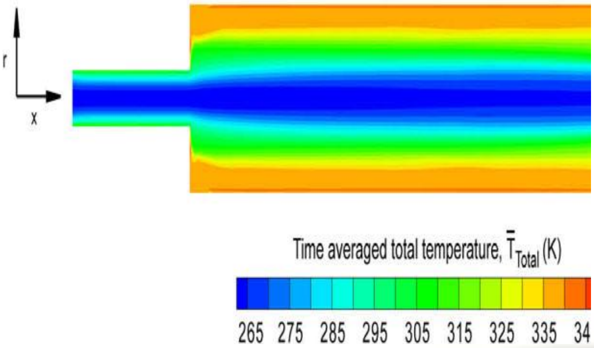

Fig. 1. Time averaged total temperature contours for the vortex tube in the r–x plane (case 1). Time averaging was performed between 0.4 and 0.8 s

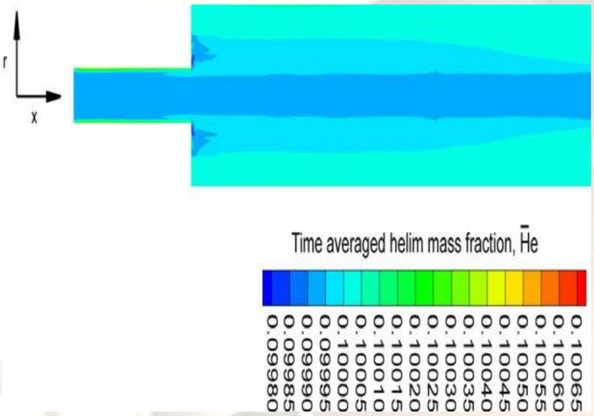

Fig. 2. Time averaged helium mass fraction contours for the vortex tube in the r–x plane (case 1). Time averaging was performed between 0.4 and 0.8 s.

S. Eiamsa-ard [8] was carried out Experimental investigation of energy separation in a counter-flow Ranque–Hilsch vortex tube with multiple inlet snail entries. The energy/
Temperature separation phenomenon and cooling efficiency characteristics in a counter-flow Ranque–Hilsch vortex tube (RHVT) are experimentally studied. The ascertainment focuses on the effects of the multiple inlet snail entries (N=1 to 4 nozzles), cold orifice diameter ratios (d/D=0.3 to 0.7) and inlet pressures (Pi=2.0 and 3.0 bar). The experiments using the conventional tangential nozzles (N=4), are also performed for comparison. The experimental results reveal that the RHVT with the snail entry provides greater cold air temperature reduction and cooling efficiency than those offered by the RHVT with the conventional tangential inlet nozzle under the same cold mass fraction and supply inlet pressure. The increase in the nozzle number and the supply pressure leads to the rise of the swirl/vortex intensity and thus the energy separation in the tube.

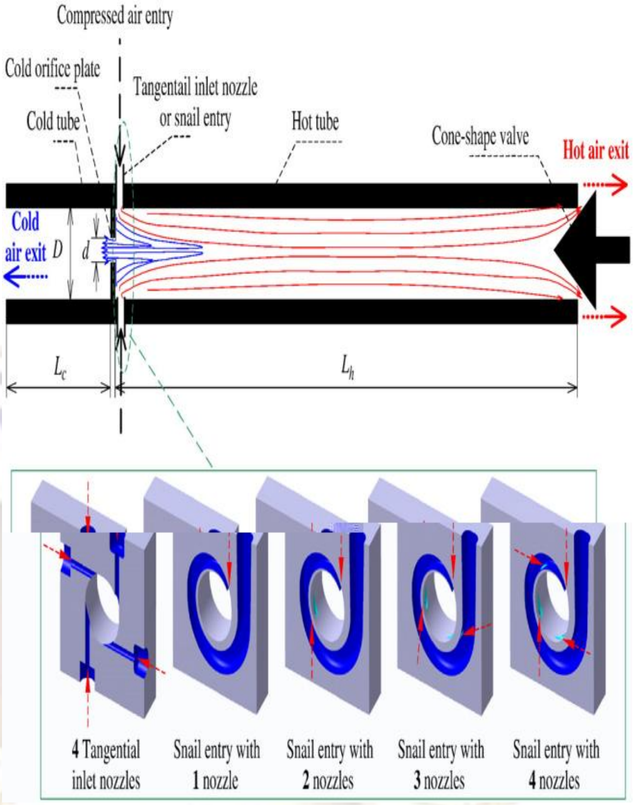

Fig. 1. Schematic diagram of a counter-flow Ranque–Hilsch vortex tube.

K. Dincer et al, [9] were carried out Experimental investigation of performance of hot cascade type Ranque-Hilsch vortex tube and exergy analysis. In this study, three Ranque-Hilsch vortex tubes were used, which have 9 mm inside diameter and length/diameter ratio was 15. Their performances were examined as one of the classical RHVT and other was hot cascade type RHVT. Performance analysis was according to temperature difference between the hot outlet and the inlet (ΔThot). The ΔThot values of hot cascade type Ranque-Hilsch vortex tubes were greater than the ΔThot values of classical RHVT, which were determined experimentally. The total inlet exergy, total outlet exergy, total lost exergy and exergy efficiency of hot stream were investigated by using experimental data. In both the classical RHVT and hot cascade type RHVT, it was found that as fraction of cold flow increases the total lost exergy decreases. It was also found that, the hot cascade type RHVT more exergy efficiency of hot outlet than the classical RHVT. Excess ΔThot value of hot cascade type Ranque-Hilsch vortex tube causes the excess exergy efficiency of hot outlet.

H.M. Skye et al, [10] were carried out Comparison of CFD analysis to empirical data in a Commercial vortex tube. This paper presents a comparison between the performance predicted by a computational fluid dynamic (CFD) model and experimental measurements taken using a commercially available vortex tube. Specifically, the measured exit temperatures into and out of the vortex tube are compared with the CFD model. The data and the model are both verified using global mass and energy balances. The CFD model is a two-dimensional (2D) steady axisymmetric model (with swirl) that utilizes both the standard and renormalization group (RNG) k-epsilon turbulence models. While CFD has been used previously to understand the fluid behavior internal to the vortex tube, it has not been applied as a predictive model of the vortex tube in order to develop a design tool that can be used with confidence over a range of operating conditions and geometries. The objective of this paper is the demonstration of the successful use of CFD in this regard, thereby providing a powerful tool that can be used to optimize vortex tube design as well as assess its utility in the context of new applications.

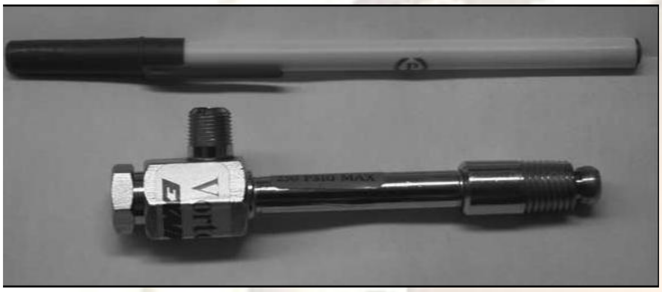

Fig. 3. Picture of vortex tube used for experiment.

IV. CONCLUSION:-

The issue concerned in all explanations is the energy transfer between different layers. When little energy is transferred from the inner flow to the outer flow, the temperature drop of cold air can be considered to be the result of sudden expansion near the entrance, and the temperature rise of hot air might be the result of friction of the multi-circulation near the hot exit. Therefore, clarification of the energy transfer between different layers is a useful approach in the investigation of the vortex tube. Clear understanding of the flow structure inside the vortex tube is required for further investigation, especially the region near the entrance in which sudden expansion is considered as the governing process and the region near the exit in which multi-circulation may be formed.

From the above study I am much more interested to work on Experimental & Numerical investigation of Ranque-Hilsch vortex tube using CFD.

V. REFERENCES

[1] Yunpeng Xue et al, were carried out "A critical review of temperature separation in a vortex tube".

[2] K. Dincer et al, were carried out "Experimental investigation of the performance of a Ranque–Hilsch Vortex tube with regard to a plug located at the hot outlet".

[3] Prof. R. K. Sahoo et al, were carried out "Numerical Analysis in Ranque-Hilsch Vortex tube."

[4] Sachin U. Nimbalkar et al, were carried out "Quantitative observations on multiple flow structures inside Ranque Hilsch vortex tube."

[5] Upendra Behera et al, were carried out "CFD analysis and experimental investigations towards optimizing the parameters of Ranque–Hilsch vortex tube."

[6] Ahmet Murat Pinar et al, were carried out "Optimization of counter flow Ranque–Hilsch vortex tube performance using Taguchi method."

[7] Tanvir Farouk et al, were carried out "Simulation of gas species and temperature separation in the counter-flow Ranque–Hilsch vortex tube using the large eddy simulation technique."

[8]. S.Eiamsa-ard was carried out "Experimental investigation of energy separation in a Counter- flow Ranque–Hilsch vortex tube with multiple inlet snail entries."

[9] .K. Dincer et al, were carried out "Experimental investigation of performance of hot cascade type Ranque-Hilsch vortex tube and exergy analysis."

[10]. H.M. Skye et al, were carried out Comparison of CFD analysis to empirical data in a Commercial vortex tube.