IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE)
e-ISSN: 2278-1684,p-ISSN: 2320-334X
PP. 89-93
www.iosrjournals.org

CFD Anyalysis &Optimization of Counterflow Ranque-Hilsch Vortex Tube A Review.

Prof. O. M. Kshirsagar[1], Ajit M. Lande[2], Akshay A. Shelar[3], Pramod S. Mane[4], Mandar G. Kurkure[5]
[1]Asst. Prof. Department of Mechanical Engineering
.[2,3,4,5]Student Department of Mechanical Engineering
JSPM Narhe Technical Campus, Pune,
Maharashtra, India 411041.

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]*

Keywords*:* *Vortextube, Ranquehilsch experiment performance parameter, Piping geometry, Turbulence modeling,Thermal Separation, CFD.*

I. INTRODUCTION

The vortex tube was invented by a French physicist named George J. Ranque in 1931 when was studying a process in a dust separated cycle there is no single theory. That explain the radial temperature separation. The vortex tube contain the following parts one inlet nozzle, cold end orifice hot end valve and a tube. Ranque, A French metallurgist and physicist about 1930 1st observed the vortex tube effect. He formed a small company to exploit the item but it soon failed it presented paper on the vortex tube to scientific society in France in 1933, but it was met with disbelieve and disinterest. There after the vortex tube disappeared for several Air until Rudolph hilsch studied it an published in findings in the mid1940.

The computational fluid dynamics (CFD) is a branch of fluid mechanics that uses numerical analysis and data structure to solved and analyze problems that involve fluid flow. Computers are used to perform calculation required to simulate interaction of liquid and gases with surfaces define the boundary conations. With high speed super computer, better solution can be achieved. Ongoing research yields software that improves the accuracy and speed of complex simulation scenarios such as a transonic or turbulent flows. Initial experimental validation of such software is performed using a wind tunnel with the final validation in full scale testing.

In the computational modeling of turbulent flow, one common objective is to obtain a model that can predict quantities n of interest such as fluid velocity, for using engineering designs of the system being modeled. For turbulence flows, the range of length scale and complexity of the phenomenon involving turbulence make most modeling approaches prohibitively expensive. The turbulence models can be classified based on computational expense.

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).

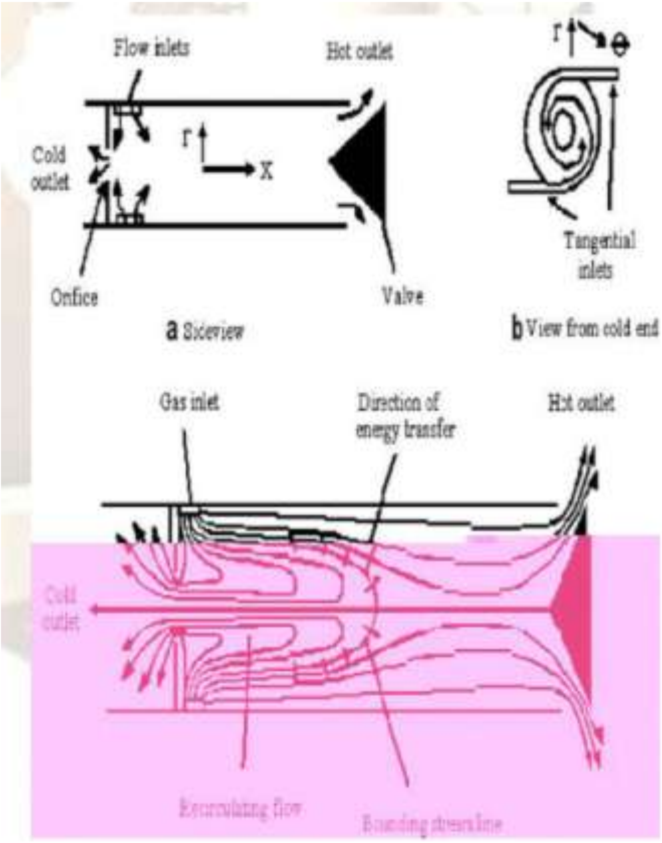

Fig. 1 – Schematic diagram of the counter flow RHVT (Cockerill, 1995).

II. METHDOLOGY

In all of the approaches the same basic procedure is followed. During preprocessing, the geometry and physical bond of the problem can be defined using computer aided design (CAD). Form there, data can be suitably processed and the fluied volume is extracted. The volume occupied by the fluied is divided into discrete cells the mesh may be uniform or non-uniform, structure or nonstructural, consisting of the combination of hexahedral, tetrahedral, prismatic. The physical modeling- for example the equation of fluid motion + enthalpy + radiation + species conversions. Boundary conditions are defined. This involving the fluid behavior and the properties at all bounding surfaces of the fluied domain. For transient problems, the initial condition are also defined. The simulation is started and the equation are solved iteratively as a stedy state or transient. Finally post processor is used for the analysis and visualization of the resulting solution.

III. PROBLEM IDENTIFIED

By going through the detailed study of various authors mentioned in the literature review it is cleared that the mechanism producing the temperature difference phenomenon in the vortex tube is not fully understood yet. Therefore, the experimentation and the CFD analysis is the only way to analyze the performance of vortex tube. Aim of project is to analyze and find the effect of different geometrical parameter of vortex tube and inlet pressure on the performance of vortex tube with the help of CFD.

IV. LITRETURE RIVIEW

Upendra Beheraetal, [1] 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.

Sachin U. Nimbalkaret al, [2]Energy separation and energy flux separation efficiencies are defined and used to recover characteristicproperties of the vortex tube. These are used to show an appropriate scale to non-dimensionalize the energy separation effect. The experimental results indicate that there is an optimum diameter of cold end orifice for achieving maximum energy separation. The results also show that the maximum value of energy separation was always reachable at a 60% cold fraction irrespective of the orifice diameter and the inlet pressure. The results are compared with the previous studies on internal flowstructure, and optimal operating parameters are shown to be consistent with a matching of orificesize with the secondary circulation being observed.

Ahmet Murat Pinar et al, [3] 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, [4] 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.

S. Eiamsa-ard [5] 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.

K. Dincer et al, [6] 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, [7] 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 andexperimental 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.

V. ADVANTAGE

- It only uses air as a refrigerant, so there is no leakage problem.
- Vortex tube is simple in design and it avoids control system.
- There is complete absence of moving part in vortex tube.
- It is light in weight and required less space.
- Cheaper in initial cost and its working expenses are also less where compressed air readily available.
- Maintenance is simple and no expert attention is required .
- No electricity or chemicals employed.
- A great time reduction and cost reduction in new design.
- The level of detailed is practically unlimited.

VI. LIMITATION

- 1.Its low thermal efficiency is mean limiting factor for its application
- 2.A vortex tube uses Compressed air as a power source.
- 3.Also the noise and availability of compressed gas may limit its application

VII. APPLICATIONS

Vortex Tubes are constructed of **stainless steel.** The wear resistance of stainless steel, as well as its resistance to corrosion and oxidation, assures that Vortex Tubes will provide years of reliable, maintenance-free operation.

- Cooling electronic controls
- Cooling machining operations
- Cooling CCTV cameras
- Setting hot melts
- Cooling soldered parts
- Cooling gas samples
- Electronic component cooling
- Cooling heat seals
- Cooling environmental chambers

VIII. CONCLUSION

The experimental and CFD work conducted shows the following results to determine the cold air temperatureand the cold air mass flow rate of the vortex tube:

1. The CFD analysis is an extremely useful tool in the design and analysis of Vortex tube.
2. The CFD enables the complete view of the cold air temperature of the vortex tube.
3. The CFD analysis shows the temperature at the core region of the cold orifice as below 0° C with different combinations of inlet air pressure, inlet nozzles and cold end orifice.

REFERENCES

[1] UpendraBehera et al, were carried out ―CFD analysis and experimental investigations towards optimizing the parameters of Ranque–Hilsch vortex tube.‖

[2] Sachin U. Nimbalkar et al, were carried out ―Quantitative observations on multiple flow structures inside RanqueHilsch vortex tube.‖

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

[4] 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.‖

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

[6] K. Dincer et al, were carried out ―Experimental investigation of performance of hot cascade type Ranque- Hilsch vortex tube and exergy analysis.‖
[7] H.M. Skye et al, were carried out Comparison of CFD analysis to empirical data in a Commercial vortex tube
[8] YunpengXue et al, were carried out ―A critical review of temperature separation in a vortex tube‖.
[9] 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‖.
[10] Prof. R. K. Sahoo et al, were carried out ―Numerical Analysis in Ranque-Hilsch Vortex tube.