ISSN: 2278-0181
Vol. 2 Issue 1, January- 2013

Exprimental Investigation & Numerical Analysis Of Ranque Hilsch Vortextube

Rahul B Patel [a], Dr. V. N. Bartaria[b]

Abstract-

Experimental studies and Computational fluid dynamics (CFD) are conducted towards the optimization of the Ranque–Hilsch vortex tubes. First experimental reading validated in ANSYS CFX 12.0 version. Then Different numbers 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.

Keywords: Ranque–Hilsch vortex tube; ANSYS CFX 12.0 simulation; Experimental validation; geometrical parameter. Thermal Performance.

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. In present study counter flow vortex tube is used which is shown in fig.1[1]

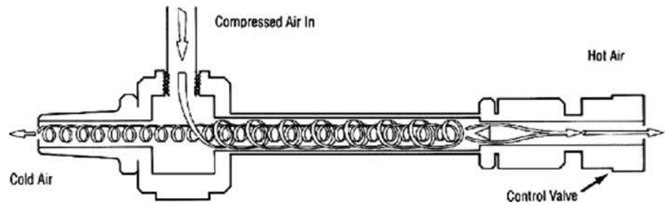

Fig. 1 – Schematic diagram of the counter flow RHVT

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).Here for modeling solid works 2009 software used.

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[2] ,no of 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 ANSYS CFX 12.0 to simulate the phenomenon of flow pattern, thermal separation, pressure gradient etc. so that they are comparable with the experimental results.

II. Experimental apparatus

Vortex tubes are classified into two groups according to their flow characteristics: counter flow and parallel flow RHVTs.In this study, a counter flow RHVT has been used.

The schematic of the experimental apparatus and measuring devices which is used for the determination of the energy separation in a vortex tube is shown in Fig.5.2. Compressed air is passing through the inlet and then it is conducted tangentially into the vortex tube. The compressed air expands

ISSN: 2278-0181
Vol. 2 Issue 1, January- 2013

in the vortex tube meanwhile is divided into cold and hot streams. The cold air leaves near the entrance nozzle, while the hot air discharges the periphery at the far end of the tube. The control valve (needle valve) may control the flow rate of the hot air. The temperature of the leaving cold and hot air in the vortex tube measure by thermometer. The pressure of inlet air is measured by pressure gauge (P) and the temperature of inlet air is measured by thermometer. [3]

The experimental tests of the vortex tube were performed with the variation of the pressure of the inlet air, Pi.

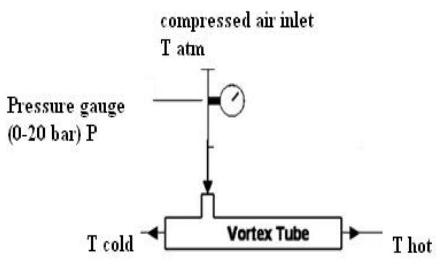

Fig 1: - schematic diagram of the experimental apparatus of vortex tube

III Modelling and Simulation:

Basics of CFD and steps for the numerical simulation of the problem were discussed in previous chapter. This chapter presents the detailed methodology adopted to carry out the numerical simulation of Vortex tube using ANSYS CFX 12.0 version. and then after the simulation work has been performed on the CFX ANSYS 12.0 version. The simulation is carried out on L/D ratio and different holes in nozzle (generator) of vortex tube.

The modeling has been performed on the Solid works 2009 version .When a parent features are modified its child features are automatically modified. It is therefore essential to reference feature dimensions. So the design modifications are correctly propagated through a model.[5]

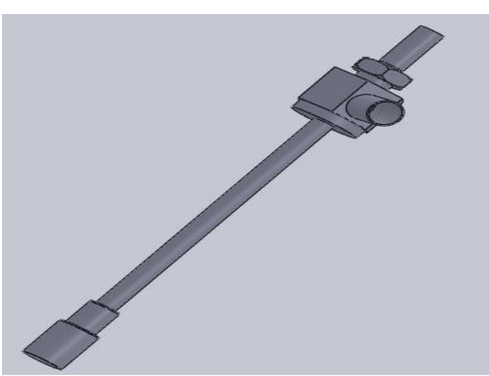

Fig 2 Isometric Model of Vortex tube

Mathematical Model :-

CFD is a numerical technique to obtain an approximate solution numerically. We have to use a discretisation method, which approximate the deferential equations by a system of algebraic equations, which can then be solved on a computer. The approximations are applied to small domain in space and/or time so that the numerical solution provides results at discrete locations in space and/or time.
The physical aspect of any fluid flow is governed by the following three fundamental
Principles:
a. Conservation of Mass
b. Conservation of Momentum
c. Conservation of Energy
These fundamental principles can be expressed in terms of partial deferential equations. CFD is a numerical technique to replace these partial deferential equations of fluid flow into the algebraic equations by numbers and discretizing them in space and/or time domain. With the advent of high speed digital computers, CFD has become a powerful tool to predict flow characteristics in various problems in an economical way.

Turbulence Models:-

Turbulence consists of fluctuation in the flow field in time and space. It is a complex process, mainly because it is three dimensional, unsteady and consists of many scales. It can have a significant effect on the characteristic of the flow. Turbulence occurs when inertia force in the flow become significant compared to viscous forces, and is characterized by a high Reynolds number. In general, the Navier-Stokes equations describe both laminar and turbulent flows without the need for additional information. However, turbulent flows at realistic Reynolds numbers span a large range of turbulent length and time scales, and Should be generally involves length scales much smaller than the smallest finite volume mesh, which can be practically used in a numerical analysis. The Direct Numerical Simulation (DNS) of these flows require computing power which is many orders of magnitude higher than available in foreseeable future.
Turbulence models are used to predict the effects of turbulence in fluid flow without resolving all scales of the smallest turbulent fluctuations. A number of models have been developed that can be used to approximate turbulence based on Reynolds Averaged Navier-Stokes (RANS) equation. Some have very specific applications, while others can be applied to a wider class of flows with a reasonable degree of confidence. One of the main problems in turbulent modeling is the accurate prediction of flow separation under adverse pressure gradient conditions. This is an important phenomenon in many technical applications, particularly for airplane aerodynamics since the characteristics of a plane are controlled by the flow separation from the wing.
In general, turbulence models based on the equation predict the onset of separation too late and under predict the amount of separation later on. This is problematic, as this behavior gives an overly optimistic performance characteristic for an airfoil. The prediction is therefore not on the conservative

side from an engineering stand-point. The models developed to solve this problem have shown a significantly more accurate prediction of separation in a number of test cases and in industrial applications. Separation prediction is important in many technical applications both for internal and external flows.

Choosing a turbulence model

It is an unfortunate fact that no single turbulence model is universally accepted as being superior for all classes of problems. The choice of turbulence model will depend on considerations such as the physics encompassed in the flow, the established practice for a specific class of problem, the level of accuracy required, the available computational resources, and the amount of time available for the simulation. To make the most appropriate choice of model for particular application, one needs to understand the capabilities and limitations of the various options. While it is impossible to state categorically which model is best for a specific application.

Many investigators have applied different turbulence models for the simulation of vortex tube.. In the present work, K-ε turbulence model is used to consider the turbulence effect.[4]

K-ε turbulence model :

One of the most prominent turbulence models, the k-ε (k-epsilon) model, has been implemented in most general purpose CFD codes and is considered the industry standard model. It has proven to be stable and numerically robust and has a well established regime of predictive capability. For general purpose simulation, the model offers a good compromise in term of accuracy and robustness.

Within ANSYS-CFX, the k-ε turbulence uses the scalable wall function approach to improve robustness and accuracy when the near wall mesh is very fine. The scalable wall functions allow simulation on arbitrarily fine near wall grids, which is a significant improvement over standard wall functions. While standard two equation models provide good predictions for many flow of engineering interest k is the turbulence kinetic energy and is defined as the variance of the fluctuations in velocity. ε is the turbulence eddy dissipation (the rate at which the velocity fluctuations dissipate) and has dimensions of per unit time. The k-ε model introduces two new variables into the system of equations.

The turbulence kinetic energy, k, and its rate of dissipation, **ε**, are obtained from the following transport equations:

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

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

Where,

$C_{1\varepsilon}$, $C_{2\varepsilon}$, $C_{3\varepsilon}$ are Constants

σ_k, $\sigma_\varepsilon$areturbulent Prandtl numbers for k and ε respectively

S_k and S_ε are user-defined source terms

The turbulent (or eddy) viscosity, μ_t, is computed by combining

k and **ε** as follows:

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

The values of model constants are:

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

k-ω turbulence model :

One of the advantages of the k-ω formulation is the near wall treatment for low Reynolds number computations. The model does not involves the complex non linear damping functions required for the k-ω model and is therefore more accurate and more robust. The models assume that the turbulence viscosity is linked to the turbulence kinetic energy and turbulent frequency via the relation.

The one-equation model is given by the following equation:

$$\frac{\partial\hat{v}}{\partial t} + u_j\frac{\partial\hat{v}}{\partial x_j} = c_{b1}(1 - f_{t2})\hat{s}\hat{v} - \left[c_{w1}f_w - \frac{c_{b1}}{k^2}f_{t2}\right](\hat{v}/d)^2 + \frac{1}{\sigma}[\frac{\partial}{\partial x_j}\left((v + \hat{v})\frac{\partial\hat{v}}{\partial x_j}\right) + c_{b2}\frac{\partial\hat{v}}{\partial x_j}\frac{\partial\hat{v}}{\partial x_j}]$$

And the turbulent eddy viscosity is computed from

$$\mu_{t=\rho\hat{v}}f_{v1}$$

Where $f_{v1=}\frac{x^3}{x^3} + \frac{x^3}{c_{v1}^3}x = \frac{\hat{v}}{v}$

SIMULATION PROCEDURE :

Following flowchart describes the general methodology adopted for solving problem of vortex tube.[6]

a. Model the Geometry

The 3D geometry of the Vortex tube was created in solid works 2009.Create cavity domain for analysis.

b. Generate Mesh

Import cavity model into ANSYS Workbench Mesh Module. A grid was generated within the flow domain. Unstructured Grid was used for meshing.

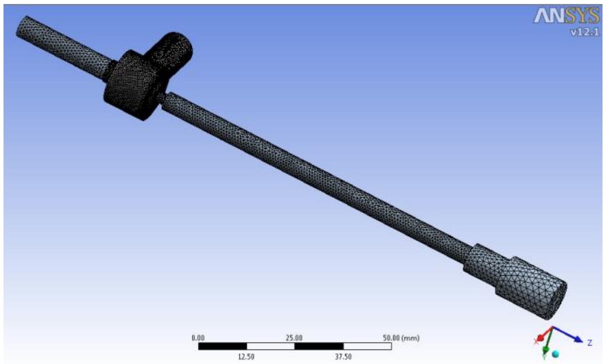

Fig 3 Meshing geometry of vortex tube

c. Specify the Initial Conditions
- Free stream conditions
- Steady state flow

d. Specify Fluid Domain for Air.
Domain Type: - Fluid
Domain Material: - Air Ideal Gas
Domain Motion: - Stationary

e. Specify the Boundary Conditions
The pressure and temperature data obtained from the experiments are supplied as input for the analysis. The boundary conditions given to simulate the vortex tube phenomenon at different regions are as follows:
1. Inlet temperature 300 K.
2. Outlet Pressure is Atmospheric 1.01325 bar

f. Define Solid Domain for Nozzle.
Domain Type: - Solid Domain
Domain Material: - Copper
Domain Motion: - Stationary

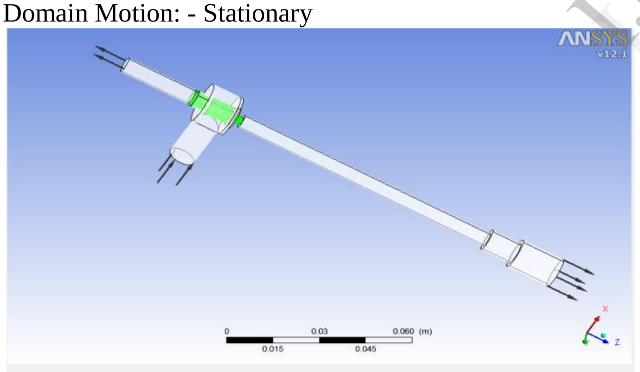

Fig 4 Solid domain for Nozzle

g. Examine and Process the CFD Results
View flow properties (Contours plot of Static pressure, pressure coefficient, Velocity vectors and path line)

IV. Comparison of Experimental and CFD Analysis of vortex tube:

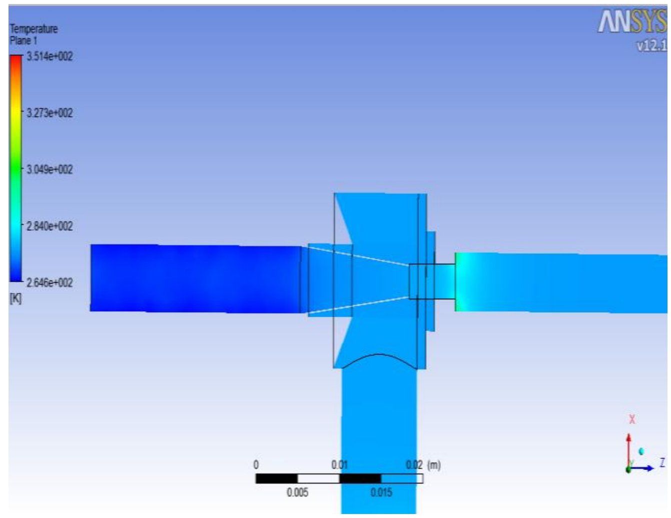

Fig 5 : Temperature Contour

Fig 2 show the temperature contour of vortex tube.

Experimental and ANSYS CFX result shown in Table-1.

Table-1

	Experimental	CFD	% Difference
Maximum Hot Gas Temperature	356 K	351.4 K	1.3
Minimum Cold Gas Temperature	289 K	264.6 K	8.44

From table- 1 maximum hot gas temperature and minimum cold gas temperature percentage difference are 1.3 % and 8.44 % respectively. Which show validation of experimental reading by analysis in ANSYS CFX 12.0

V. CFD analysis towards optimization of parameters :

1.Number Of Nozzles holes:

Table-2

No. of Nozzle Holes	Temperature Difference between Hot and Cold End
1	18.4 K
2	19.6 K
4	22.6 K
6	25.6 K
8	23.4 K
10	19.2 K

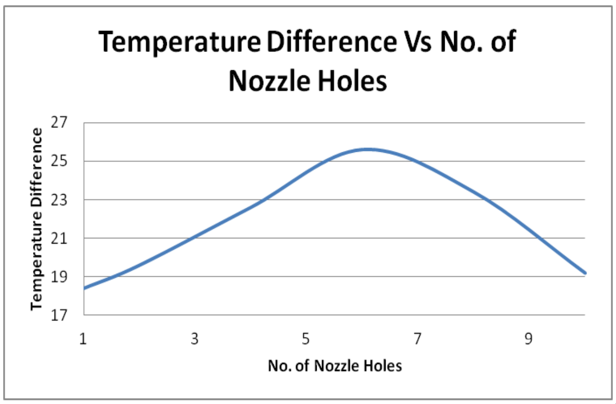

Fig 6 Temperature difference V/S No of nozzle hole

Figure 6 shows, increase the nozzle hole from 1 to 10 then at six number of hole temperature difference become maximum. After that increase number of hole temperature difference is decrease.

2.L/D ratio:

Table-3

L/D	Temperature Difference
10	86.8 K
15	94 K
20	98 K
25	103 K
30	109 K
35	104 K

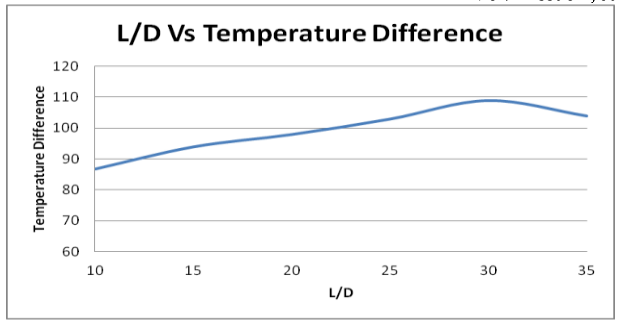

Fig 7 L/D V/S Temperature difference

Figure 7 shows, by increasing length to diameter ratio temperature difference increase. But after L/D ratio 30 it will decrease.

3. Inlet Pressure :

Table-4

Inlet Pressure (bar)	Temperature Difference
4	72 K
6	86.8 K
8	94 K

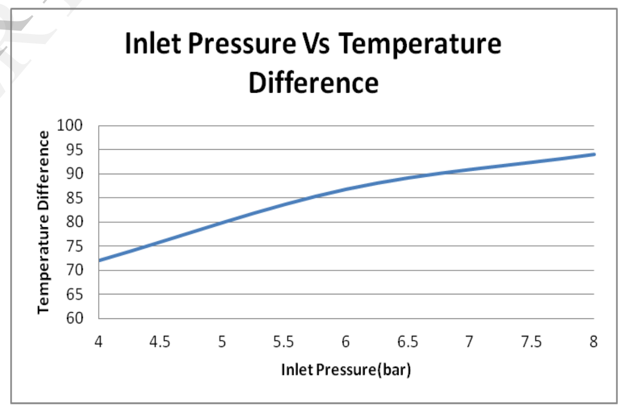

Fig 8 Inlet pressure V/S Temperature Difference

Figure 8 shows, by increasing inlet pressure the temperature difference increase.

VI. Conclusion

In this paper the Experimental study on energy separation in the vortex tub. Also studied governing of geometrical parameters on the performance characteristics of the vortex tube are investigated by the ANSYS CFX 12.0. The result is that the optimum value of L/D is determined. The L/D ratio for best performance is 30. Number of nozzle hole is also determined. The six number of hole gives maximum temperature difference. The cold air temperature difference increases by increasing the inlet pressure, meanwhile there is an optimum efficiency at a specific inlet pressure.

ISSN: 2278-0181
Vol. 2 Issue 1, January- 2013

VII. REFERENCES

[1] M.H. Saidi et al, were carried out "Experimental modeling of vortex tube refrigerator". Applied Thermal Engineering 23 (2003) 1971–1980.

[2] W. Frohlingsdorf, H. Unger, Numerical investigations of the compressible flow and the energy separation in Ranque–Hilsch vortex tube, Int. J. Heat Mass Transfer 42 (1999) 415–422.

[3] M.H. Saidi,20 april 2003 were carried out "Experimental modeling of vortex tube refrigerator" page:- 1971-1980 science direct. Applied thermal engineering.

[4] J. Ferziger and M. Peric, Computational Methods for Fluid Dynamics, Third edition-revised, 2004.

[5]Solid work 2009.

[6] ANSYS CFX 12.0 version.