iJRASET
International Journal For Research in
Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 2 **Issue:** X **Month of publication:** October 2014

DOI:

International Journal for Research in Applied Science & Engineering Technology(IJRASET)

CFD Analysis of Vortex Tube for Various Cross Sectional Nozzles

Naresh Kumar[1], Anandkumar S Malipatil [2]

Department of Thermal Power Engineering, VTU-Gulbarga

Abstract-- In this article computational fluid dynamic analysis of a 3-D steady state compressible and turbulent flow has been carried out through a vortex tube. The numerical models use the standard realizable k-ε turbulence model to stimulate the vortex tube domain along with boundary conditions. This article has focused on the energy separation and flow field behavior of vortex tube by utilizing all the cross section that is rectangular straight, rectangular helical , circular straight and circular helical nozzles have been investigated and their principal effects as cold temperature difference was compared. The studied vortex tube dimensions are kept the same for all models.
Key words: vortex tube, stagnation point, circular nozzles, straight nozzles, helical nozzles

I. INTRODUCTION

A vortex tube is a Thermo-fluidic device which generates cold and hot stream with single injection of pressurized gas. The mechanism of vortex tube is as shown in fig 1

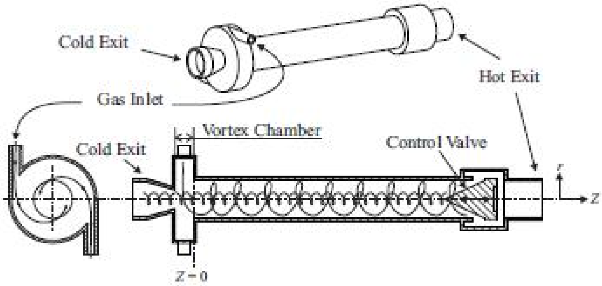


Figure 1: The Ranque-Hilsch vortex tube.

R Liew, J C H Zeegers , J G M Kuerten and W R Michalek [1] experiment on Vortex tube for the measurement of Temperature, Pressure and Velocity. Results show that the cooling power is larger than the heating power due to a heat loss to the surroundings. The velocities were obtained by means of Laser Doppler Anemometry. W. Rattanongphisat, S.B Riffat and G. Gan [2] studied three dimensional models to stimulate the physical behavior of the flow such as temperature and the pressure inside the vortex tube. H.M Skye, G.F. Nellis, and S.A Klein [3] present a comparison between the performance predicted by a CFD model and experimental measurement taken using a commercially available vortex tube. The data and the model are both verified using global mass and energy balances. The CFD model is 2D steady axisymmetric model that utilizes both standard and renormalization group (RNG).

Recent efforts that have successfully benefited of CFD could explain the basic principles behind the energy separation produced by the vortex tube. More designing parameters such as tube length and its geometry, cold and hot exit area, number of nozzles can be governed by the flow field behavior in a vortex tube. But among them, nozzle geometrical shape is a specific case because it can be significantly enhanced the entrance gas velocity to vortex chamber. The present investigation therefore tends to explore the effect of helical nozzle geometry as a one of the main fundamentals of vortex tube structure in describing energy separation.

II. VORTEX TUBE MODEL DESCRIPTION

The CFD models of present research are based on the analysis of Skye et al. [3] experimental vortex tube. The vortex tube had been equipped with 6 straight nozzle. But in this will go with 3 nozzles keeping the inlet area same for all models. Since high rotating flow inside the vortex tube makes a complex compressible turbulent flow.

Hence this article has given a direction to study the effect of both straight and helical nozzle. In the new regarding, the Skye's vortex tube is modeled numerically with respect to 3 rectangular straight , 3 circular straight , 3 rectangular helical and 3 circular helical instead of 6 straight nozzles such that the total nozzle area are kept constant to all set of nozzles. This is due to the fact that this article believes that helical nozzles can play very considerable role in appropriately operating of a vortex tube even for a few number of nozzles in comparison with straight nozzles.

Basic assumptions for all computations of the vortex tube flows were made as follows: A circumferential pressurized gas inlet and two axial orifices for cold and hot stream with air as a working fluid. Since the chamber consists of 3 slots, the CFD models are assumed to be rotational flow. Dimensional geometric details of these models are presented in table 1.

Measurements	Skye's experimental vortex tube	Present vortex tube with 3 number of either straight or helical nozzle
Working tube length	106mm	106mm
Working tube internal diameter	11.4mm	11.4mm
Nozzle height	0.97mm	0.97mm
Nozzle width	1.41mm	2.82mm
Nozzle total area	8.2mm2	8.2mm2
Cold exit diameter	6.2mm	6.2mm
Hot exit area	95mm2	95mm2

Table 1: Geometric summery of CFD models used for vortex tube.

III. GOVERNING EQUATION

CFD is based on the fundamental governing equation of fluid dynamics which expresses the fundamental physical principal of fluid dynamics The assessment has been carried out on an air-cooled finned-tube condenser of a vapour compression cycle for air conditioning system.

The partial differential equation form of the continuity equation

$$\frac{\partial(\overrightarrow{\rho v})}{\partial t} + \vec{\nabla}(\rho\vec{v}\vec{v}) = \vec{\nabla}.\rho + \vec{\nabla}.\vec{\tau} + \rho.\vec{b}$$

Where ρ is the density of fluid (kg/m3); t is the time (sec); v is the velocity vector (m/s).

The conservation form, partial differential equation called Navier-Stokes equation.

$$F=m.a$$

Where F is body force (N), m is mass (kg), and a is the acceleration (m/s2)

$$\frac{\partial\left(\rho\vec{v}\right)}{\partial t} + \vec{\nabla}.\left(\rho\vec{v}\vec{v}\right) = \vec{\nabla}.p + \vec{\nabla}.\vec{\tau} + \rho\vec{b}$$

The conservation form, partial differential equation is shown below.

$$\frac{\partial\rho e}{\partial t} + \vec{\nabla}.\left(\rho e\vec{v}\right) = \rho\dot{q} - \vec{\nabla}.\left(k\vec{\nabla}T\right) - \vec{\nabla}.\left(p\vec{v}\right) + \vec{\nabla}.\left(\vec{\vec{\tau}}.\vec{v}\right) + \rho\vec{b}.\vec{v}$$

Where v is the rate of volumetric heat addition per unit mass, T is temperature; e is the internal energy per unit mass.

IV. BOUNDARY CONDITIONS

Mesh Model;

Solver type: Pressure based

Problem model: Standard realizable k-ε.

Material;

Fluid: Ideal Gas

Boundary Conditions;

Total mass flow inlet: 8.35gm/s

Temperature: 294.2 K

Wall condition: Stationary and no slip condition

Outlet: Pressure outlet for both.

International Journal for Research in Applied Science & Engineering Technology(IJRASET)

Standard realizable k-ε model is used for all type of model analysis. Air (ideal gas) is used as a working medium. At the nozzle inlet, mass flow inlet condition is given for all model analysis and pressure outlet boundary conditions were given for both hot outlet and cold outlet as 20kpa and 15kpa respectively. Assuming wall should be stationary and no-slip condition for all models.

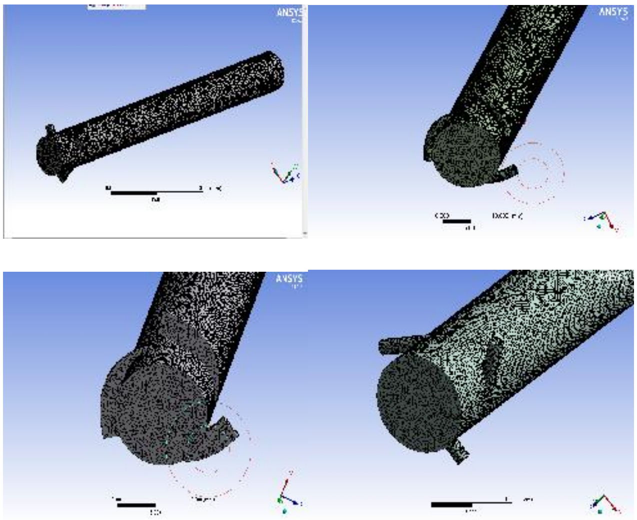


Figure 2:CFD models of various vortex tubes.(a)Rectangular straight (b) circular helical © Rectangular helical (d) circular straight

V. VALIDATION

A compressible form of the Navier- stokes equation together with appropriate k-ε turbulence model are derived and solved by using the ANSYS FLUENT software. In order to discretise the derivative terms, the second order upwind and quick schemes are employed to momentum, turbulence and energy equations. The temperature profile along the length of the vortex tube was obtained from the present calculations was compared with the experimental results.

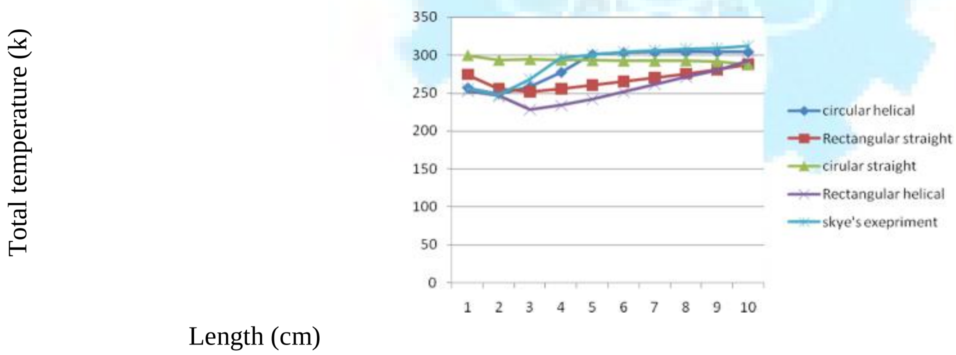
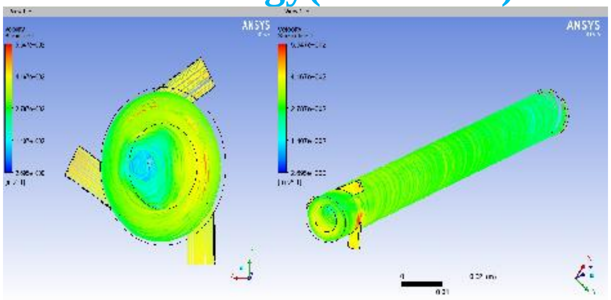


Figure 3: Temperature comparison of CFD models Vs Experimental model

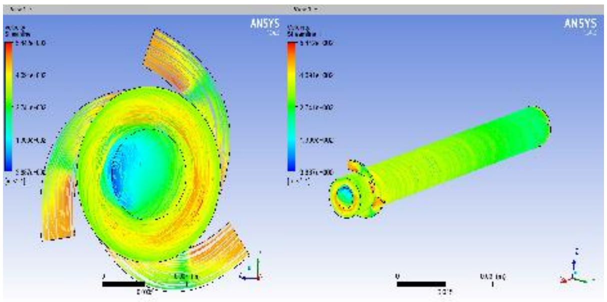
VI. RESULT AND DISCUSSIONS

The flow pattern at the vortex chamber of the CFD models of vortex tube, as the velocity fields are shown in figure 4. Indeed, vortex chamber is a place that, cold exit is completely coincided to the end plane of its, but with smaller diameter than the main tube. In figure 4, locally injected momentum by means of nozzle into vortex chamber is restricted to nozzle exit area only that is instantaneously and set reasonable is only the creation of a symmetric flow field.

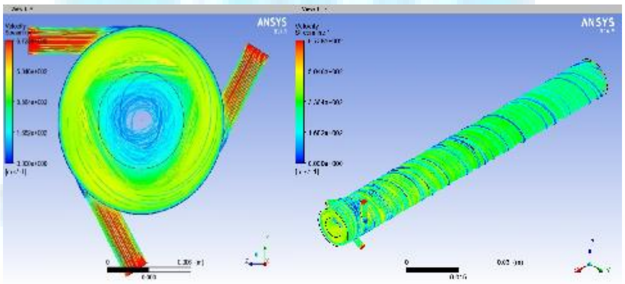
From figure 4, objection of locally momentum injection is recovered by increasing of nozzle width (nozzle area) because total nozzle area is constant for all of nozzles set. This situation caused a uniformly injection of momentum to produce semi continues high momentum zones in the rotating flow domain; as u can seen in figure by red areas. So the exit momentum from each nozzle is more effective to move downstream flow towards next nozzle.



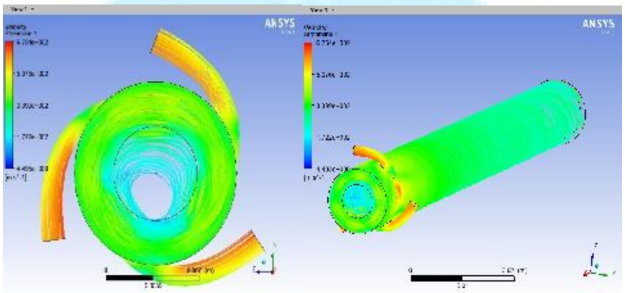
(a)



(b)



(c)



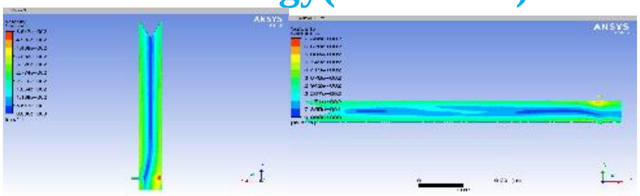
(d)

Figure 4: Stream lines for various CFD models (a)Rectangular straight (b) Rectangular helical (c) Circular straight (d) Circular helical

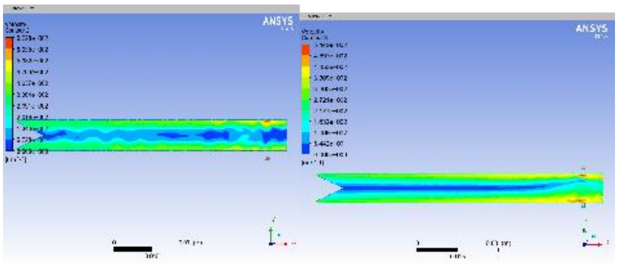
A. Velocity and Total temperature

The properly exit velocity has provided a reasonable and interested rotating flow so that each nozzle gains sufficient enough energy to the downstream flow to push toward the next nozzle. These types of nozzles show that they can produce somewhat higher velocity than others. Thus it is a criterion to attain maximum cold temperature difference in the vortex tube.

ISSN: 2321-9653
International Journal for Research in Applied Science & Engineering Technology(IJRASET)



(a) (b)

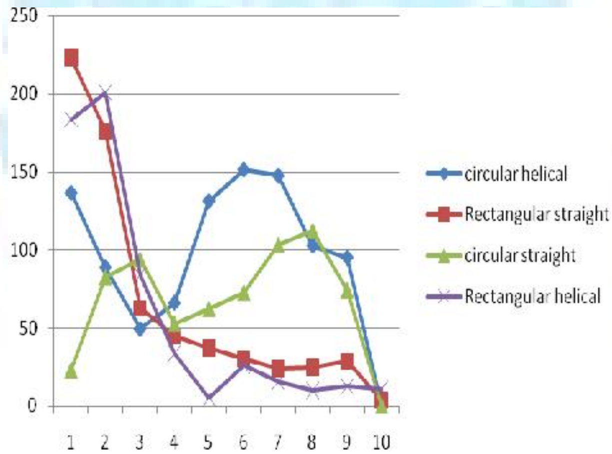


(c) (d)

Figure 5: velocity contours for various vortex tube (a) Rectangular straight (b) Circular helical (c) Circular straight (d) Rectangular helical.

Comparing the velocity components one can observe that the velocity has a greater magnitude in helical nozzle than the straight nozzle. The radial profile of the swirl velocity indicates a free vortex near the wall and becomes another type of vortex, namely forced vortex at the core which is negligibly small.

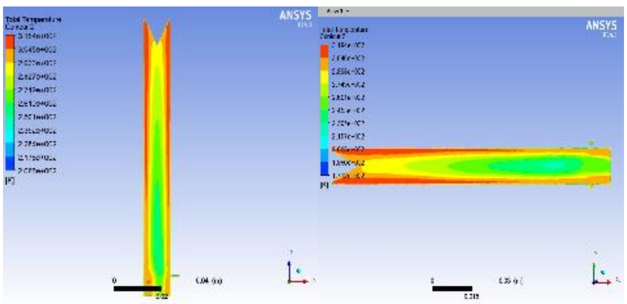
Velocity (m/s)



Length (cm)

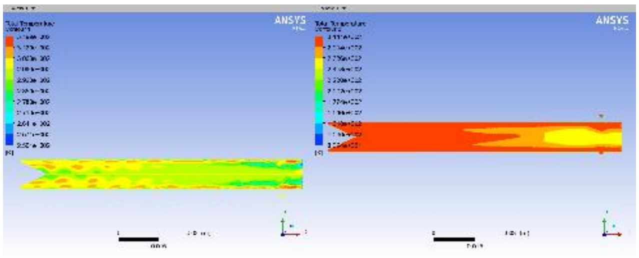
Figure 6: velocity at the mid Vs length for various vortex tubes.

The above figure shows the radial profiles of the velocity at the mid plane along the length of the vortex tube for various nozzles as shown.



(a) (b)

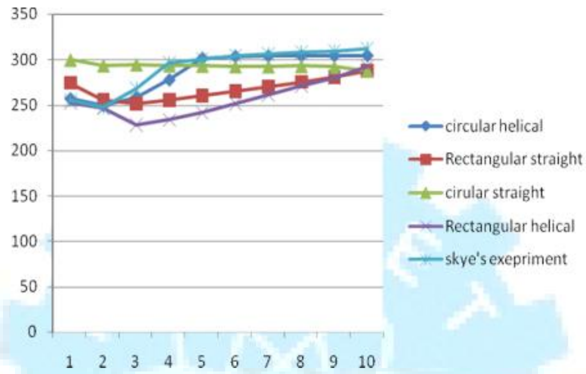
ISSN: 2321-9653
International Journal for Research in Applied Science & Engineering Technology(IJRASET)



(b) (d)

Figure 7: Temperature distribution at the mid plane vs. length for various nozzles (a) Rectangular straight (b) Rectangular helical (c) Circular straight (d) Circular helical.

Total temperature (k)



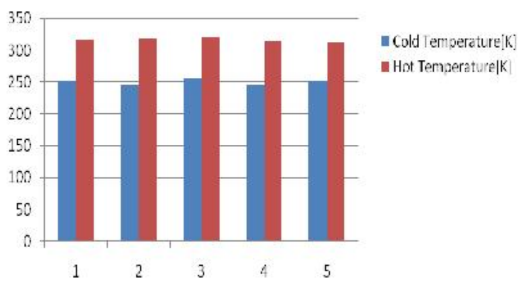
Length (cm)

To attain maximum swirl velocity and maximum cold temperature difference velocity distribution together with wall temperature location also would be the important parameters in designing a good vortex tube. The results show that the positions of stagnation point for all of models are too closer to the hot exit. But helical nozzle set causes the position of this point drawn rather little to the hot exit end. One can consider due to somewhat closeness of separation point of helical nozzle set to the hot exit, it brings maximum cold temperature difference in vortex tube.

The CFD model described thus far is compared to the measured data gathered from the commercial vortex tube. Emphasis is placed on temperature and energy separation. Measurement of pressure are also reported, but rigorously used for model verification. In the CFD model, the cold exit pressure boundary was specified until the experimentally measured cold fraction was achieved.

The comparison between model and experiment described above were based on the measured boundary conditions and geometric characteristics. Of these measurements, the use of measured inlet area provides the largest source of error. Altering the inlet area of the CFD model is significantly affected the total pressure of the incoming compressed air. Because the total inlet pressure is a main factor contributing the magnitude of power separation within the vortex tube, the inlet area was changed to increase the CFD total pressure to values closer to the experimentally measured inlet pressures.

1-Rectangular straight
2-Rectangular helical
3-Circular Straight
4-Circular helical
5- Experimental



The above figure shows the histrogram of various vortex tube with different cross sections and experimenatal vortex tube. Such that the temperature difference were nearly equal to the experimental values.

Table 2: comparison of temperature difference of different vortex tube

Model	Cold exit temp. [K]	Hot exit temp [K]	ΔTi,c [K]	ΔTi,h[K]	ΔTc,h [K]
Rectangular straight	250.106	315.452	44.094	21.252	65.346
Rectangular helical	245.336	319.432	48.864	25.232	74.096
Circular straight	257.124	319.928	37.076	25.728	62.804
Circular helical	245.876	314.125	48.324	19.925	68.249
Skye's experimental	250.24	311.5	43.96	17.3	61.26

VII. CONCLUSIONS

A computational approach has been carried out to realize the effects of nozzle shape.

- Higher swirl velocity can effectively influence the exit cold gas temperature.
- Helical nozzles are suitable to the desired amount of energy separation and higher cold gas temperature difference.
- Helical shape nozzles gain sufficient energy to the downstream flow in order to conduct them towards next nozzle.
- The total temperature separations (hot and cold exit) predicted by the CFD model of both straight and helical nozzles were found to be in good agreement with available experimental data.

REFERENCES

[1] R Liew, J C H Zeegers, J G M Kuerten and W R Michalek."Temperature, pressure and velocity measurement on the Ranque Hilsch vortex tube". 6th European thermal sciences conferences (Eurotherm 2012), IOP Publishing-2012.

[2] H.M. Skye , G.F. Nellis, S.A. Klein. "Comparison of CFD analysis to empirical data in a commercial vortex tube. International Journal of refrigeration 29 (2006) pg 71-80.

[3] Upendra Behera, P.J.Paul, S. Kasthurirengan, R. Karunanithi, S.N Ram, K. Dinesh, S. Jacob. "CFD analysis and experimental investigations towards optimizing the parameters of Ranque-Hilsch vortex tube. International journal of Heat and mass transfer , 48 (2005),3 pp.442-450.

[4] H.R Thakare, Y.R. Patl and A.D.Parekh. "A review of computational studies of temperature separation mechanism in vortex tube". Bonfring international journal of Industrial engineering and management science, Vol.3, No. 2, June 2013.

[5] N.Pouramahmoud, A.Jahangiramini, A.Izadi. "Optimization of low pressure vortex tube via different axial angles of injection nozzles". International journal of engineering. Vol. 26, No.10, (October 2013) 1255-1266.

[6] Gupta U.S, Joshi M.K and Pawar C.B. "Experimental performance evaluation of counter flow vortex tube". Journal of environmental Research and development. Vol. 7 No.1A, July-September 2012.

[7] G. Maruthi Prasad Yadav, K. Nagalakshmi, Dr. P. Mallikarjuna Reddy, Dr. B. Uma Maheshwar. "Effect pf hot end obstruction and nozzle on the performance on vortex tube. Int. journal of engineering research and application ISSN:2248-9622, Vol.3, Issue 5, Sep-Oct 2013, pp.1792-1796.