Ijesm
Consulting, help, relaxation INTERNATIONAL JOURNAL OF ENGINEERING SCIENCES & MANAGEMENT

PERFORMANCES ANALYSIS OF VORTEX TUBE ON CFD WITH STRAIGHT AND HELICAL NOZZLE

Prabhat Kaushal[1]*, Sohail Bux[2] and Amitesh Paul[3]
1Research Scholar, 2Professor, 1,2Department of Mechanical Engineering , Agnos College of Technology, Bhopal, Madhya Pradesh, INDIA
3Principal, Institute of Polytechnic Engineering, Bhopal, Madhya Pradesh, INDIA
Email ID: prabhatkaushal87@gmail.com, +918120766507

ABSTRACT

A vortex tube is a simple energy separating device which splits a compressed air stream into a cold and hot stream without any no moving parts, external energy supply or chemical reactions. This paper focuses on the effect of the size of hot nozzle on the performance of the Ranque–Hilsch vortex tube. The present research has focused on the energy separation and flow field behavior of a vortex tube by utilizing both straight and helical nozzles. Basically three kinds of nozzles set include, 3 and 6 straight types and 3 helical type's nozzles have been investigated and their principal effect on cold temperature difference was compared. All vortex tubes dimensions are kept the same for models. All these performances analysis is done on three dimensional flow domain using Computational Fluid Dynamics (CFD).

Keywords: Vortex Tube, Swirl velocity, helical nozzles, Straight nozzles, CFD.

INTRODUCTION

Vortex tube is a simple device, which can cause separation of energy. The principle of a swirl tube is produced by the tangential velocity of the main driving force for separating energy in the vortex tube. It consists of the nozzle, the swirl chamber, the cold plate separation, the hot water valve and the hot and cold end, no moving parts. In the vortex tube, in which the work, the compressed air expands in the nozzle and enters the vortex tube tangentially at high speed, mass vortex, the air inlet is divided into low pressure and flow of, hot and cold temperatures, one of which, the peripheral air has a higher temperature than the initial air, while the other, the central flow has a lower temperature.

The vortex tube was discovered in 1930 by Georges Ranque, a student of French physicist, he experimented with a vortex pump that had developed type when they realized expelling hot air from one end and cold the other air. Ranque soon forgot about his pump and started a small business to exploit the commercial potential for this strange device that produces hot and cold air with no moving parts. However, it soon failed and the vortex tube slipped into obscurity until 1945when Hilsch Rudolph, a German physicist, published a widely read scientific paper on the device. Vortex was the first company to develop this phenomenon into practical and effective solutions for the application of industrial refrigeration.

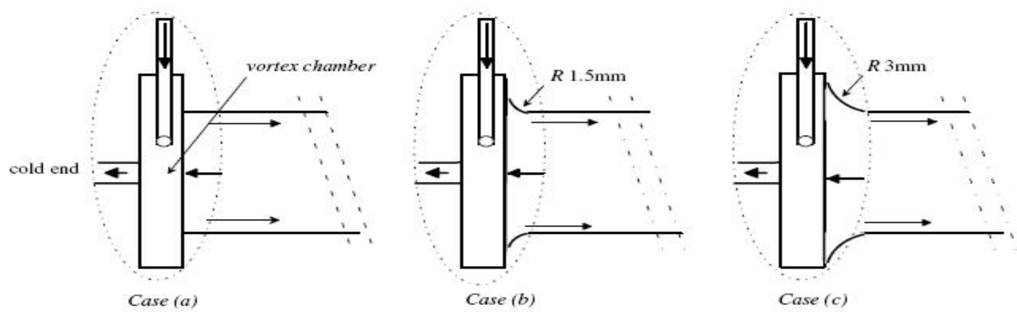
Much Earlier, the great nineteenth century physicist James Clerk Maxwell postulated that since heat involves the movement of molecules, we might one day be reliable to get hot and cold air of the same device using a "friendly little demon" that sort of molecules and separate hot and cold air Thus, the vortex tube has-been variously Known as "Ranque Vortex Tube", the "Hilsch tube", the "Tube-Ranque Hilsch" and "Maxwell Demon". By any name, it has in recent years Gained acceptance as a simple, reliable and low cost for a wide variety of Industrial cooling problems on site. When it was Announced that the device using the air flow discharged hot and cold at the same time at both ends, many people thought that this is a case of violation of the second law of thermodynamics. Later, Eckert gave a theoretical explanation of His behavior. Since Then, the tube had been the subject of numerous articles and research projects. But market acceptance has been slow. Today, there are more than 300,000 Estimated an vortex tubes are used commercially worldwide.

LITERATURE SURVEY

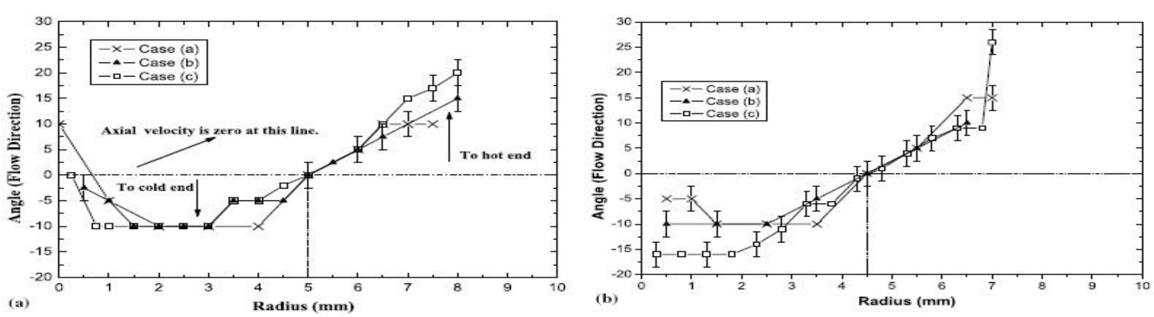
Some experimental and theoretical work on vortex tube has been done in the last decades. Both the industrial and academic people have taken interest in this area. The following is a review of the research that has been completed especially in vortex tube. The literature survey is arranged according to similarity to the work done in this thesis.

Ahlborn et al. [1] that the presence of a secondary flow field contributes to the migration of energy in the RHVT. The hypothesis states that a primary fluid stream consisting of outer and inner vortices exist that extends the length of the tube , and a secondary flow of the loop which converts heat between two flows of the vortex, which acts as agent cooling of an open thermodynamic cycle.

Gao et al. (2004) [2] experimented with three different entrance conditions: Without rounding off the entrance, rounding off radius=1.5mm, rounding off radius=3.0mm and at 2 different stations: Z/D =1.47 and Z/D =2.97. Temperature is measured by chromel - constantan thermocouple. Comparison was made with results of other authors due to similar measuring techniques. Fig. shows that the flow angle shoots up near the wall. It is inferred from the results that the rounding of entrance length can decrease secondary reverse flow in the core.



Three entrance conditions



Comparison of flow directions at (a) Z/D=1.47 (b) Z/D=2.97

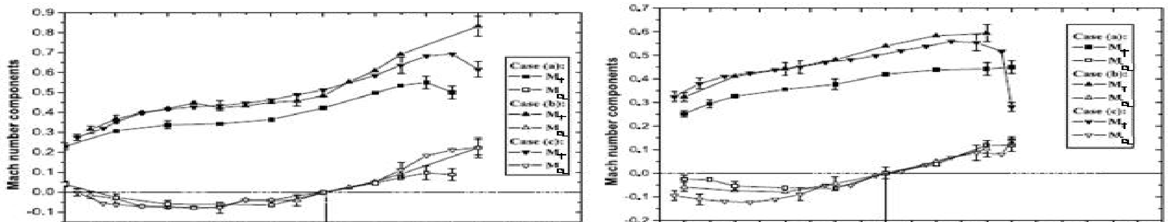
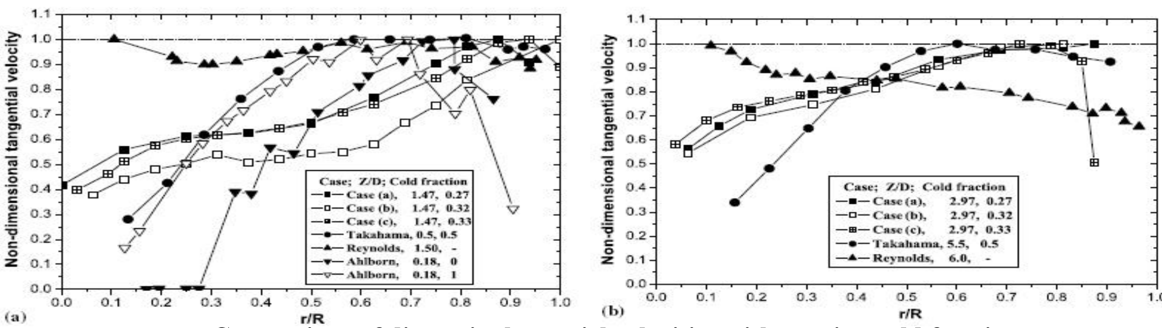


Fig. 2.3 Static and Stagnation temperatures in (a) Z/D=1.47 (b) Z/D=2.97

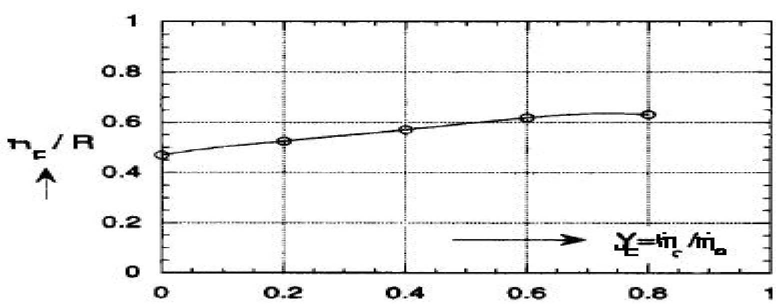


Comparison of dimensionless swirl velocities with varying cold fractions

Reynolds results seem exceptional since swirl velocity is nonzero only after r=0.3R as seen in Fig 2.4. Considering the small drops in swirl velocity at the wall, it seems that the boundary layer is very small.

Ahlborn (Ahlborn and Groves, 1997) [1] from above Fig. show that when the cold fraction is very low, there is no flow in r<0.3R region and when it is very high, the swirl velocity drops very fast near wall at the station Z/D=0.18. The results show a higher axial velocity than swirl velocity near the wall. It is conjectured that different sizes of the vortex tube, different numbers of injection nozzles and different sizes of probes are the main reasons resulting in different results.

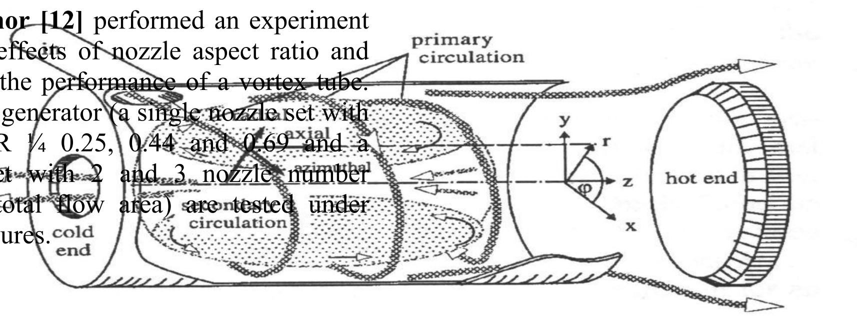
He also found the variation of critical radius with the cold fraction, as shown in Fig. 14. It is observed that rc increases with yc. This also means that the axial velocity curves shift upwards. The presence of a backflow core at yc=0 when the cold end is closed is unexpected.



Variation of rc with yc

Gao et al. [2] ascertained the presence of this secondary flow by using apparatus such as that shown in Fig. 2-1 and by measuring the axial and tangential velocities in the RHVT. The recirculation region of flow was discovered when it was established that the mass flow returning towards the cold outlet is much larger than the cold mass flow emerging out the cold end.

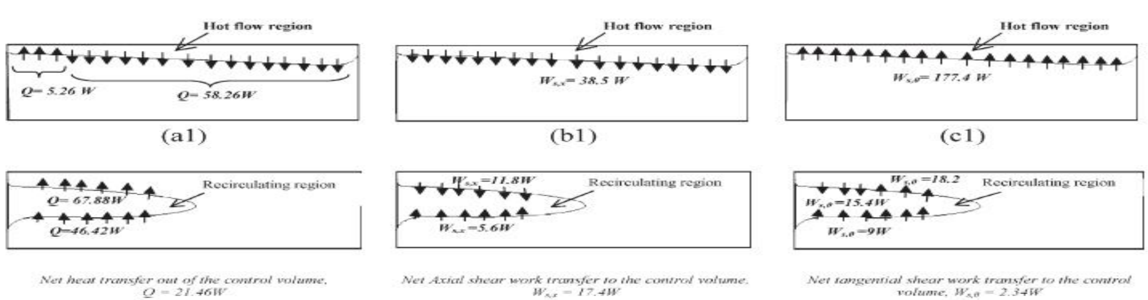
According to author [12] performed an experiment to investigate the effects of nozzle aspect ratio and nozzle number on the performance of a vortex tube. Two sets of vortex generator (a single nozzle set with aspect ratio of AR ¼ 0.25, 0.44 and 0.69 and a multiple nozzle set with 2 and 3 nozzle number having the same total flow area) are tested under different inlet pressures.



They used dry air as the working fluid. The experimental results reveal that the nozzle aspect ratio has a great effect on the energy/temperature separation mechanism. By the experiment they found that

- The temperature difference is increasing with increasing inlet pressure.
- The performance of the vortex tube decreases with increasing nozzle aspect ratio.
- The single nozzle yields to better performance than the vortex tube with 2 and 3 nozzles.
- The optimum cold mass fraction varies with the change of nozzle number.

Aljuwayhel et al. (2005) [3] divided the flow region into three instead of two control volumes included the re-circulating region as a control volume as well. The effect of the re- circulating region is that it takes away some cold elements of the cold region and mixes them with hot region to bring its temperature down. Similarly some hot elements get mixed with the cold flow and bring the temperatures up. It increases the thermal resistance of the heat transfer path and the reduced terminal temperature difference reduces and overall heat transfer reduces.



Direction and magnitude of heat, axial shear work and tangential shear work transfer

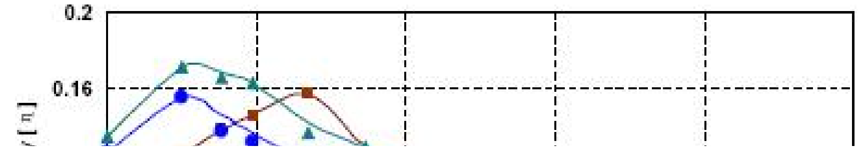
Behera et al. [4] have previously published literature on a RHVT using a complete three dimensional vortex tube and suggest that secondary flow may be a performance degradation mechanism and is best avoided.

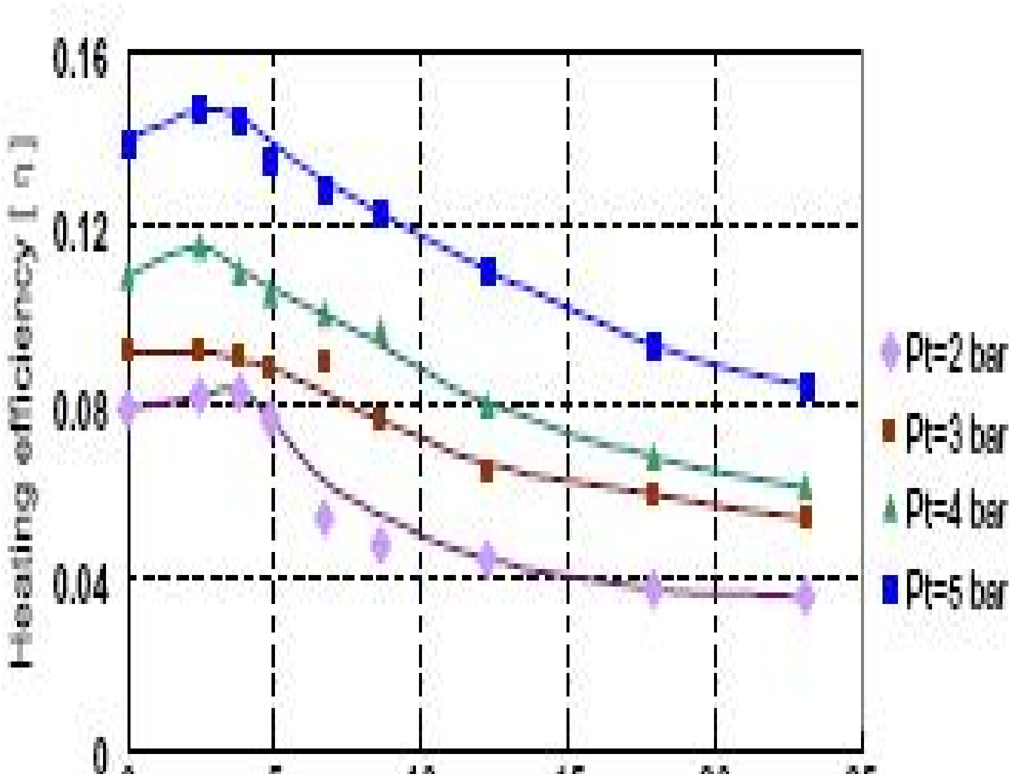
Aydin (Aydin and Baki (2005)) [6] nitrogen to be the optimal gas as compared to oxygen and air. It is observed (see Table 6) that lower the molecular weight of the gas, greater is the temperature difference. Also cold temperature difference was found to increase by increasing the specific heat capacity ratio. This is seen from the fact when Saidi (Saidi and Valipour (2003)) [8] tested helium, oxygen and air (γ=1.393, 1.4 and 1.667 respectively), helium turned out to be the optimal gas. From the thermodynamic point of view, the RHVT involves a process in which we have "adiabatic expansion of a part of a gas from higher to lower pressure. The expansion work is transferred to the other part of the gas, which is simultaneously adiabatically compressed, and then after throttling to the same low pressure leaves the system" (Aydin and Baki (2005) [7]). Thus when inlet pressure increases, the driving force for expansion of the cold stream increases and this in turn help obtain greater gradients within the same geo -metrical dimensions (equation 31, 32, 33) and thus increases energy transfer. This in general increases the cold temperature difference and cooling efficiency.

Nimbalkar (Nimbalkar and Muller (2008)) [7] conducted experiments to find the optimal cold end orifice diameter. Non-dimensionalized various quantities were plotted againt cold fraction and for different orifice diameters. It was found that the maximum performance was obtained at yc=0.6 irrespective of orifice diameter and inlet pressure. For yc<0.6, the plots nearly coincide as they ideally should. For yc>0.6, the larger orifice diameter gave better efficiency. This behavior was explained by Love (1974) and Piralishvili and Fuzeeva (2005) in terms of pressure balance inside the tube. The total pressure drop inside is addition of the drops (a) at the inlet, (b) due to the vortex generator, (c) across the flow, (d) across cold end orifice and (e) across the hot fraction control valve. The pressure drop across the hot fraction control valve controls the cold fraction. For a constant inlet pressure and orifice diameter, the flow structure depends on the pressured drop-(e). Thus the cold fraction related the flow structure with performance of the tube. As cold fraction increases, the stagnation point moves towards the hot end and the radial stagnation point goes near the wall. When cold fraction reduces, the opposite occurs. It is thought that there exist critical positions of radial and axial stagnation points where efficiency is greatest. It is believed that these conditions occur at cold fraction of 0.6.

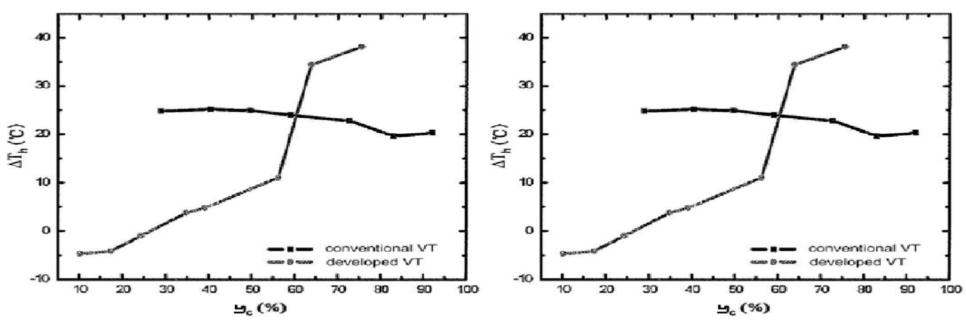
Xue (Xue and Ajormandi (2008)) [9] Different angles were set in the test to find the effect of the vortex angle. It was shown that the swirl angle has played an important role in the separation of cold and hot flows and performance of the vortex tube. A smaller vortex angle showed a difference of greater and better performance to the efficiency of heating temperature of the vortex tube. However, small angles vortices have led to greater cooling effectiveness only to lower inlet pressure values. The existence of the maximum efficiency rHVT cooling system has not been clearly determined in the investigation and research with low air pressure injected necessary. For a given swirl angle, the best performance of vortex tube can be seen in the increased inlet pressure and the highest performance can be found when the inlet pressure is about 4 bar.

Heating efficiency profile

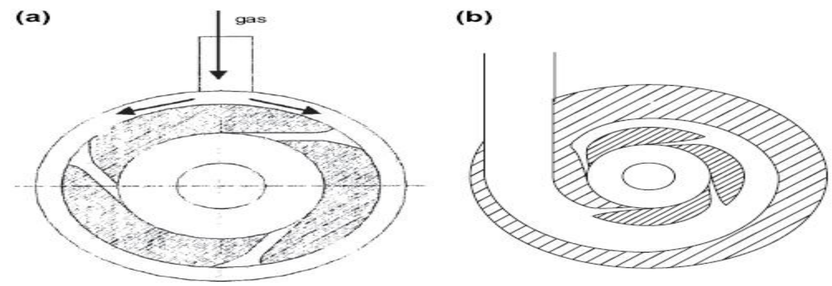




Wu et al. (2007) [10] developed innovative nozzles having the same Mach number in the section perpendicular to the axis of the nozzle. The Mach number along the axes increases in the same slope. This maintains the high peripheral speed of the section after the steam outlet. In design, there is a smooth transition from the line input to the line right ring and the passage area of the circular flow decreases so that the speed remains constant. This supports the losses due to vortex formation due to sudden changes in area and due to sudden changes in speed. A diffuser installed to reduce the peripheral speed at the hot end to zero. Diffuser reduces viscosity and residual velocity loss and helps to greatly reduce L / D. We see that the developed RHVT was better than classic cold RHVT top fractions.



Comparison between conventional and developed vortex tube



Conventional nozzle **Developed nozzle**

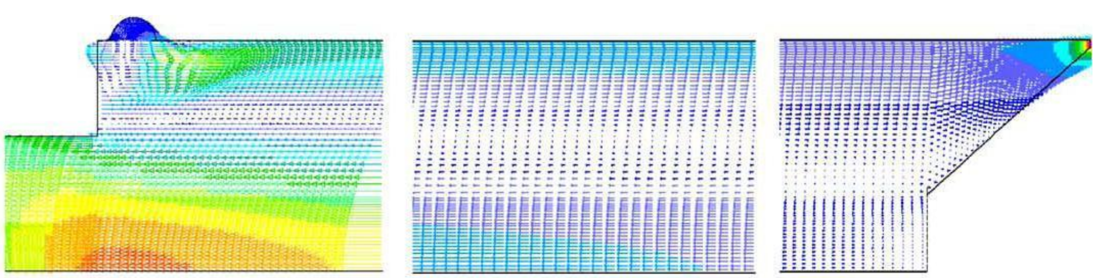
Behera et al. (2005) [5] used star-CD code to model the RHVT to optimize the number of nozzles, nozzle profile, cold end diameter, length to diameter ratio, cold and hot gas fractions, maximum COP as heat engine and as refrigerator. The meshing and analyses were carried out in P4, 2.4Ghz and 1GB RAM machine with refinement meshes near critical areas like nozzle inlet areas, hot end exit area and wall boundary. The pressure and temperature data obtained from experiments were used as input. Stagnation boundary condition was used at inlet. Reference temperature was 293 K. Numerical solution was obtained by a variant of SIMLPLE algorithm-incompressible fluid assumption.

The optimum cold end diameter for obtaining maximum hot end temperature is 7mm while that for getting minimum cold end temperature is 6mm. Lesser is the hot fraction, higher the hot end temperature. Similarly, lower the cold fraction, lower the cold end temperature. The above obtained optimized parameters were use by Behera et al. (2008) [4] to do the energy separation analysis. It was also found that temperature difference increased with higher L/D as seen in Fig.

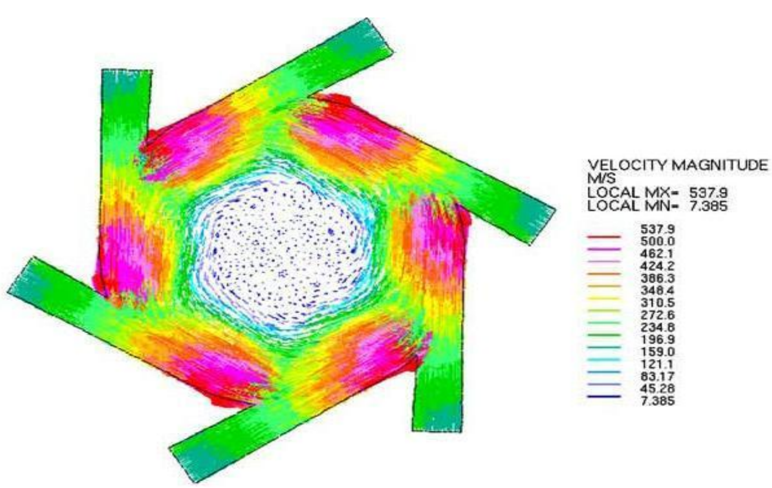
Working conditions and results of Behera et al (2005) CFD

D (mm)	Pin (MPa)	Pc (MPa)	Ttot at inlet (K)	Optimal L/D
12	0.5422	0.136	300	20-30

optimal N and nozzle profile	optimal dc	Optimal yc, yh	optimal COP as heat engine	optimal COP as refrigerator
6, convergent	7,6	0.6, 0.12-0.15	0.59	0.83



Flow pattern for 0.05m length sections at Z/L=0.0374, 0.46, 0.9



Numerical Methodology of vortex tube

The CFD models of present research are based on the analysis of Skye et al. [17] experimental vortex tube. The vortex tube had been equipped with 6 straight nozzles. In an experimental and numerical analysis process, he found a good correlation between two approaches, however their CFD model has employed 2-D computational model. Since the high rotating flow inside the vortex tube makes a complex compressible turbulent so that this system has been investigated in with respect to various geometrical parameters such as tube length. The exploration of stagnation point location showed that the experimental device tube length was just appropriately, as the numerical results prediction. Hence, this article has devoted its research direction to study effects of both nozzles number and its geometry on the mentioned device. In the new regarding, the Skye's vortex tube is modeled numerically with respect to 3 straight and 3 helical nozzles instead of 6 straight nozzles in PRO-E software such that the total nozzles 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.

The CFD simulations of vortex tube having diameter 11.4 mm and length of 106 mm. the working fluid is air. The CFD simulations were performed using commercial CFD code ANSYS FLUENT 14. The assumptions made for modeling of vortex tube are steady state, so energy equations are present. Liquid phase turbulence was modeled using the k -e model; The simulation of fluidized bed performed by solving the governing equations of mass and momentum conservation using fluent software, Eulerian multi-fluid model is adopted in the present work.

RESULTS & DISCUSSION

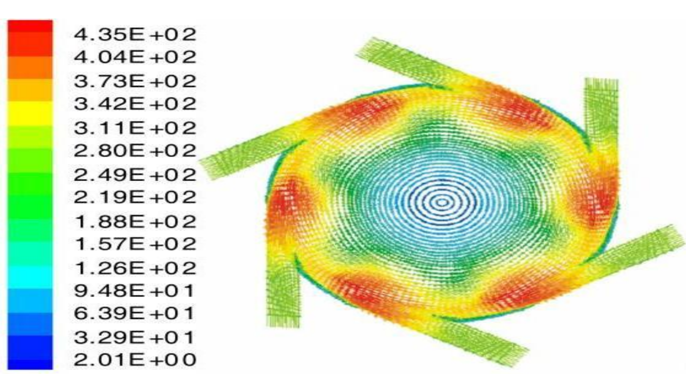
The Results and discussion of vortex tube by using Numerical simulation present in this chapter. The general procedure for the simulation using commercial CFD software package ANSYS 14 (FLUENT).

Flowchart of general procedure for the simulation using FLUENT

Geometric summery of CFD models used for vortex tube

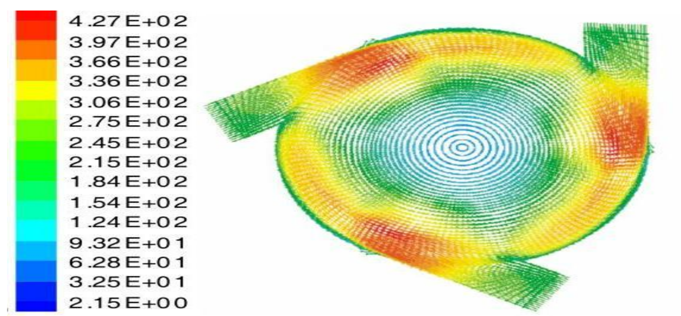
	experimental vortex tube	**number of either helical or straight nozzle**
Working tube length	106mm	106mm
Working tube internal diameter, (D)	11.4mm	11.4mm
Nozzle height (H)	0.97mm	0.97mm
Nozzle width (W)	1.41mm	2.82mm
Nozzle total inlet area	8.2mm	8.2mm
Cold exit diameter	6.2mm	6.2mm
Hot exit area	95mm	95mm

The flow patterns at the vortex chamber of the three CFD models of vortex tube, as the velocity field, are shown in fig. Indeed, vortex chamber is a place that, cold exit is completely coincided to the end plan of its, but with smaller diameter than the main tube.



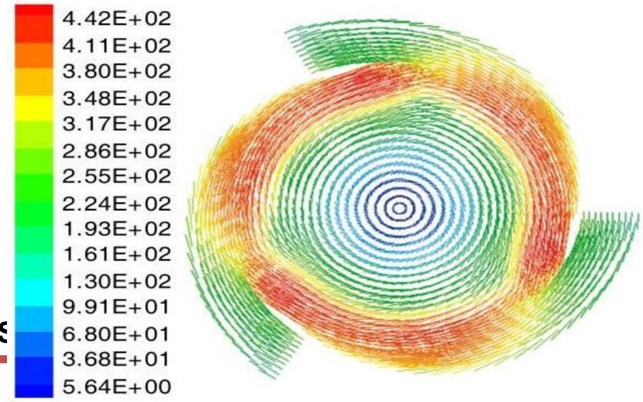
Velocity patterns at the vortex chamber obtained from CFD for: 6 straight nozzles

In above fig., in spite of 6 straight nozzles presence, locally injected momentum by means of nozzles into vortex chamber is restricted to nozzle exit area only, that is instantaneously and low order because of small width of nozzle and division of total mass among the nozzles. What makes this set reasonable is only the creation of a symmetric flow field.



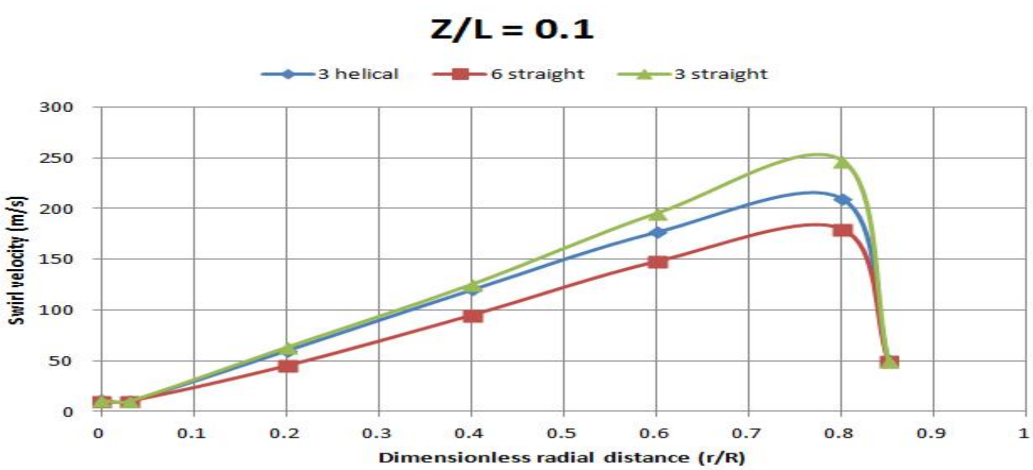
Velocity patterns at the vortex chamber obtained from CFD for: 3 straight nozzles

In above fig. objection of locally momentum injection is recovered by increasing of nozzle width (nozzle area) because total nozzles 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 can be seen in the fig. 5.4 by red areas. It must be reminded that at this condition since the nozzles number is less than the last one, so the exit momentum from each nozzle is more effective to move downstream flow toward next nozzle.



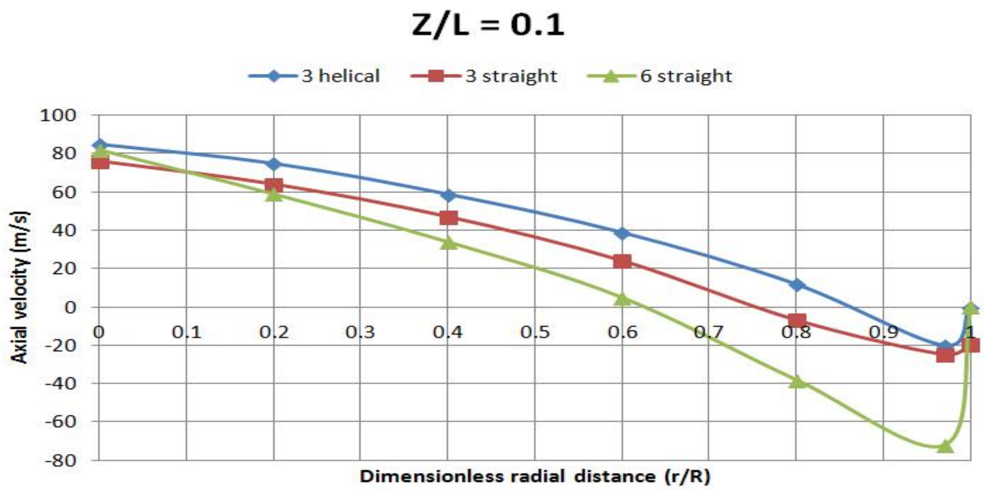
Velocity patterns at the vortex chamber obtained from CFD for: 3 helical nozzles, α = 0.3

Finally in above fig., applying of 3 helical nozzles has removed the issue of instantaneously momentum injection and semi continues high momentum zones in the vortex chamber. These are implemented by formation of good tangential exit velocity from each helical nozzle. The properly exit swirl 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 swirl velocity than the others.
Thus, it is a criterion to attain maximum cold temperature difference in the vortex tube device. It must be regard that in this condition the vortex tube has operated only with 3 helical nozzles instated of 6 straight nozzles.



Radial profiles of swirl velocity at various axial positions, α = 0.3

From above figure illustrates the radial profiles for the swirl velocity at different axial locations. Comparing the velocity components, one can observe that the swirl velocity has greater magnitude of the axial velocity. The magnitude of the swirl velocity decreases as ever moves towards the hot end exit. 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.



Radial profiles of axial velocity at various axial positions, α = 0.3

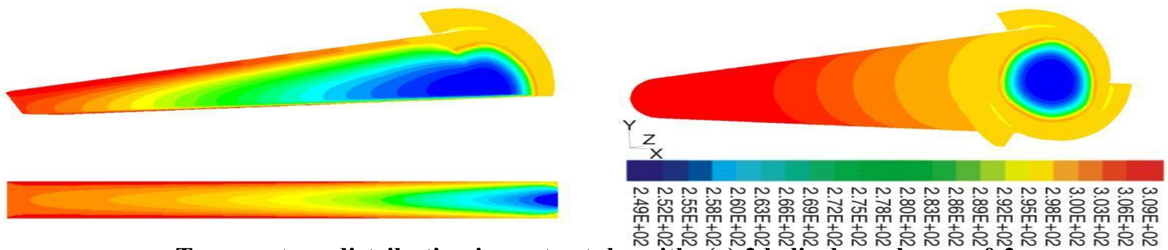
From above figure shows the radial profiles of the axial velocity magnitude at different axial locations for specified cold mass fraction equal to 0.3. At the initial distances of tube, Z/L = 0.1, cold gas has axial velocity greater than hot stream near the wall. Its maximum value occurs just in the centerline and moves towards cold exit conversely to the hot flow which leaves the tube through the hot exit. In the higher values of dimensionless length i. e. Z/L= 0.7, axial velocities of hot gas flow rises gradually in contrary to cold gas flow. The flow patterns relevant to 3 and 6 straight nozzles, have lead a cold temperature difference less than 3 helical nozzles. In comparison; however, the cold temperature difference of 3 straight nozzles has lower values than 6 straight nozzles, which would be expected. Table 5.2 summarized total temperature difference of cold and hot ends gases for various types of nozzles.

Comparison of temperature difference for vortex tube with different nozzles, α = 0.3

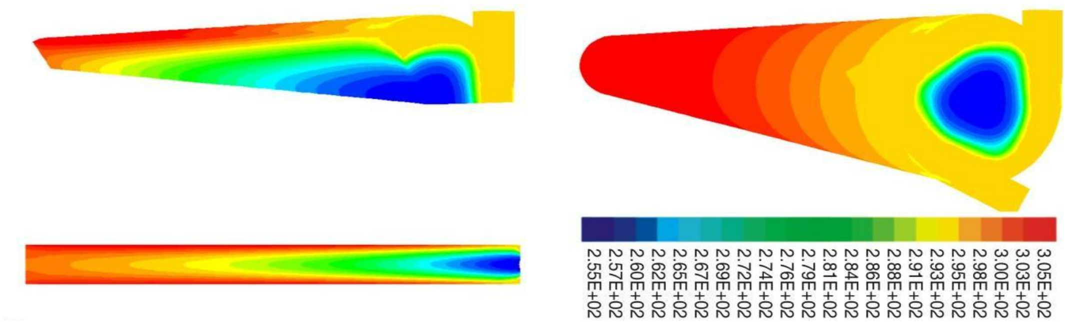
Model type	Cold exit temperature [K]	Hot exit temperature [K]	ΔT i ,c [K]	ΔT i, h [K]	ΔT c, h [K]
3 helical	249.034	309.302	45.166	15.102	60.268
3 straight	255.124	304.837	39.076	10.637	49.713
6 straight	250.24	311.5	43.96	17.3	61.26

The vortex tube with 6 straight nozzles reaches hot and total temperature difference higher than 3 straight and 3 helical nozzles. However, if only the cold temperature difference is a criterion of well operating in that machine, 3 helical nozzles will provide good cooling condition. The total temperature distribution contours obtained from CFD analysis are plotted in fig. It shows peripheral flow to be warmer and core flow colder relative to inlet temperature equals to 294.2 K, giving maximum hot gas temperature of 313.451 K and minimum cold gas temperature of 249.034 K for 3 helical nozzles.

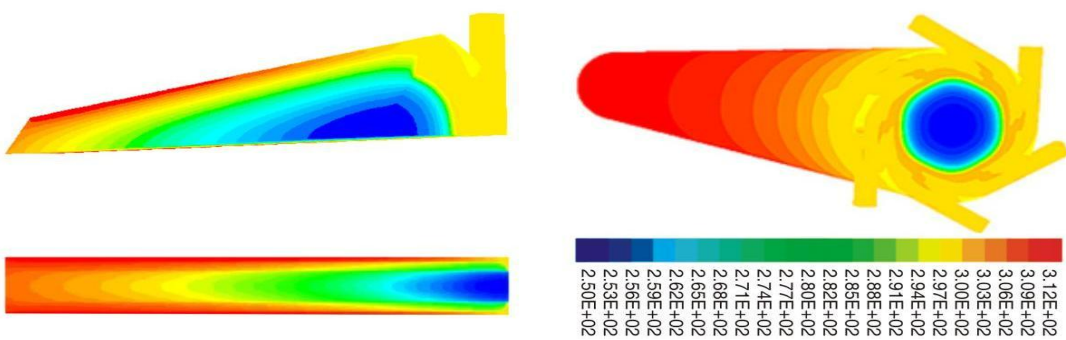
▪



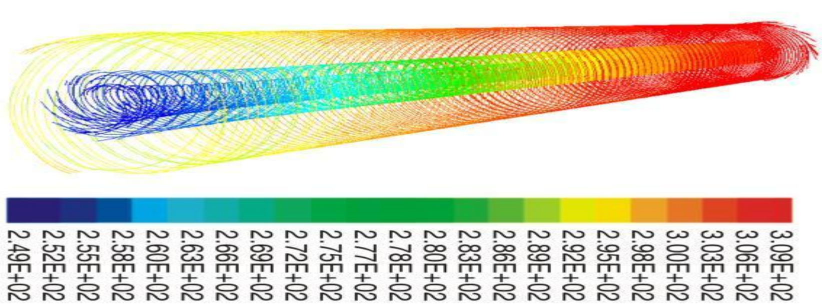
Temperature distribution in vortex tube with: (a) 3 helical nozzles, α = 0.3



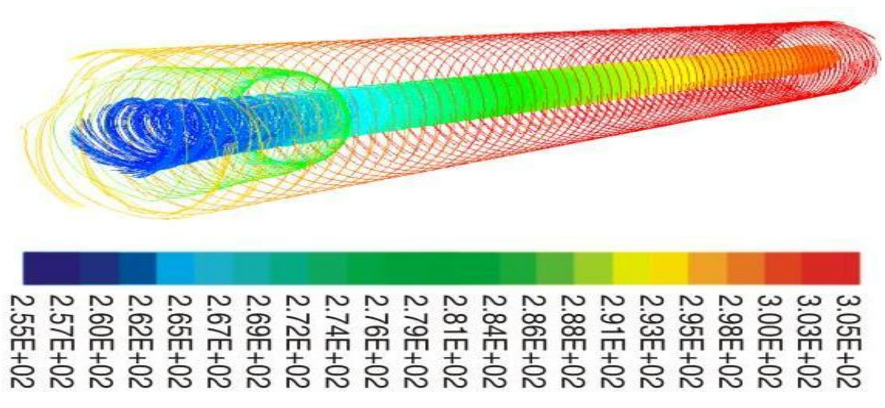
Temperature distribution in vortex tube with: (b) 3 straight nozzles, α = 0.3



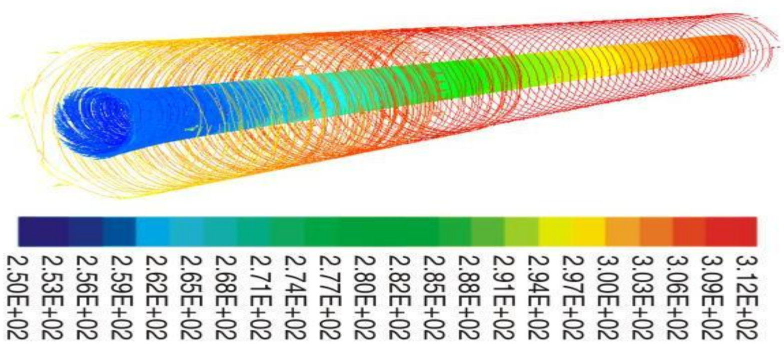
Temperature distribution in vortex tube with :(c) 6 straight nozzles, α = 0.3



3-D path lines coloured by total temperature along the vortex tube with: (a) 3 helical nozzle, α = 0.3



3-D path lines coloured by total temperature along the vortex tube with: (b) 3 straight nozzle, α = 0.3



3-D path lines colored by total temperature along the vortex tube with: (c) 6 straight Nozzles, α = 0.3

CONCLUSION

A computational approach was made to realize the effects of the form of injection nozzles and their number in the performance of the vortex tube. In a compressible flow 3 - D, the standard k turbulence model is used - and analyzing the flow patterns through the CFD modeling. Three nozzles game consists of six lines, three lines and three helical nozzles were investigated. The main objective was considered to maximum difference in cold temperature. Thus, numerical results show that the increase in the speed of swirling, due to appropriately shaped nozzles can effectively influence the temperature of the cold gas outlet. The comparison of flow fields in the three nozzle assemblies was approved helical nozzles are suitable for the desired amount of power separation and greater temperature difference of cold gas.

REFERENCES

1. Ahlborn B, Groves S. (1997). „Secondary flow in a vortex tube", Fluid Dynamics Research, 21 (1997), 73-86.
2. Gao et al., C. M., Bosschaart, K. J., Zeegers, J. C. H., de Waele, A. T. A. M. (2004). „Experimental study on a simple Ranque-Hilsch vortex tube", Cryogenics, 45 (2005), 173- 183.
3. Aljuwayhel, N. F., Nellis, G. F., Klein, S. A. (2004). „Parametric and internal study of the vortex tube using a CFD model", International Journal of Refrigeration, 28 (2005), 442-450.
4. Behera U, Paul, P. J., Dinesh K, Jacob S. (2008). „Numerical investigations on flow behavior and energy separation in Ranque-Hilsch vortex tube", International Journal of Heat and Mass Transfer, 51 (2008), 6077-6089.
5. Behera, U., and Paul, P. J., CFD Analysis and Experimental Investigations towards Optimizing the Parameters of RHVT, International Journal of Heat and Mass Transfer, Vol.48, 2005, pp. 1961-1973.
6. Aydin O, Baki M. (2006). „An experimental study on the design parameters of a counter flow vortex tube, Energy, 31 (2006), 2763-2772.
7. Nimbalkar S, Muller M. R. (2008). „An experimental investigation of the optimum geometry for the cold end orifice of a vortex tube", Applied Thermal Engineering, 29 (2009), 509-514.
8. Saidi, M.H., Valipour, M. S. (2003). „Experimental modeling of a vortex tube refrigerator", Applied Thermal Engineering, 23 (2003), 1971-1980.
9. Xue Y, Ajormandi M. (2008). „The effect of vortex angle on the efficiency of the Ranque - Hilsch vortex tube", Experimental Thermal and Fluid Science, 33 (2008), 54-57.
10. Wu, Y. T., Ding Y, Ji, Y. B., Ma, C. F., Ge, M. C. (2007). „Modification and experimental research on vortex tube", International Journal of Refrigeration, 30 (2007), 1042-1049.
11. Pinar, Ahmet. M., Uluer, O., & Kırmaci, V. (2009)." Optimization of counter flow Ranque–Hilsch vortex tube performance using Taguchi method". International Journal of Refrigeration, 32(6), 1487–1494.
12. Avcı, M. (2013). The effects of nozzle aspect ratio and nozzle number on the performance of the Ranque–Hilsch vortex tube. Applied Thermal Engineering (Vol. 50, pp. 302–308).
13. Gupta, U. S., Joshi, M. K., & Pawar, c. B. (2012). Experimental performance evaluation of counter flow vortex tube, 7(1), 496–502.
14. Yilmaz, M., Kaya, M., Karagoz, S., & Erdogan, S. (2008). A review on design criteria for vortex tubes. Heat and Mass Transfer, 45(5), 613–632. doi:10.1007/s00231-008-0447-8
15. Bramo, A. R., Pourmahmoud, N., Computational Fluid Dynamics Simulation of Length to Diameter Ratio Effect on the Energy Separation in a Vortex Tube, Thermal Science, 15 (2011), 3, pp. 833-848.
16. Aljuwayhel, N. F., Nellis, G. F., Klein, S. A., Parametric and Internal Study of the Vortex Tube Using a CFD Model, Int. J. Refrig., 28 (2005), 3, pp. 442-450
17. Skye, H. M., Nellis, G. F., Klein, S. A., Comparison of CFD Analysis to Empirical Data in a Commercial Vortex Tube. Int. J. Refrig., 29 (2006), 1, pp. 71 -80