Proceedings of the
International Conference on Mechanical Engineering 2011
(ICME2011) 18-20 December 2011, Dhaka, Bangladesh

ICME11-FL-008

COMPUTATIONAL ANALYSIS OF THE EFFECT OF VORTEX PARAMETERS IN A VORTEX TUBE

Anayet U. Patwari, **N.M Reehan**, **Adib Bin Rashid and Ashikur Rahman**

Department of Mechanical and Chemical Engineering
Islamic University of Technology (IUT), Dhaka, Bangladesh

ABSTRACT

This paper presents a finite element analysis to investigate the effect of vortex parameters in a vortex tube. In the simulation Air was taken as the fluid. A CAD model of the vortex tube was generated using AutoCAD and mesh file was created using Ansys ICEM CFD software. Simulation parameters and boundary conditions were extracted from an actual experimental condition. Simulation results were obtained as velocity vector diagram and temperature contour plot using Ansys CFX which illustrates the flow pattern and temperature gradient inside the vortex tube. This CFD simulation provides better understanding of the generation of vortex and the change of flow parameters such as velocity and temperature inside a vortex tube.

Keywords: Vortex Tube, CFD Simulation, Temperature Contour, Velocity Distribution.

1. INTRODUCTION

The vortex tube, also known as the Ranque-Hilsch vortex tube, is a very simple device in which a single stream of high pressure air is divided into two streams of low pressure air, one colder than the supply and one hotter than the supply. Within the device a stationary tangential nozzle imparts rotation to the air. That rotation produces concentric flows with same sense of rotation but with opposite pitch so that the outer flow moves toward the hot end and the inner flow moves back toward the cold end. A valve at the hot end determines the fraction of the air that will leave either end. With features as no moving parts, simplicity of manufacture and obvious applications to direct cooling or heating the success is obvious in environments with an existing compressed air source. Pressurized gas is injected tangentially into a swirl chamber and accelerates to a high rate of rotation. Due to the conical nozzle at the end of the tube, only the outer shell of the compressed gas is allowed to escape at that end. The remainder of the gas is forced to return in an inner vortex of reduced diameter within the outer vortex.

The vortex tube was invented in 1933 by French physicist Georges J. Ranque. German physicist Rudolf Hilsch improved the design and published a widely read paper in 1947 on the device, which he called a *Wirbelrohr* (literally, whirl pipe). [2] The vortex tube was used to separate gas mixtures, oxygen and nitrogen, carbon dioxide and helium, carbon dioxide and air in 1967 by Linderstrom-Lang [3]. Vortex tubes also seem to work with liquids to some extent [4]. In 1988 R.T.Balmer applied liquid water as the working medium. It was found that when the inlet pressure is high, for instance 20-50 bar, the heat energy separation process exists in incompressible (liquids) vortex flow as well. K. Stephan [5] investigated of energy separation in a vortex tube. M. Kurosaka [6] studied acoustic streaming in swirling flow inside a vortex tube. Merwin Sibulkin [7] carried out unsteady-flow analysis for the development of the flow in the vortex tube. T. Passot [8] presented direct numerical simulations of homogeneous turbulence. Also many CFD analysis [9-11] have been done by several scientists in order to optimize vortex tube and finding out the effect of parameters related to vortex performance. M. H. Saidi [12] carried out experimental analysis on vortex tube operation.

This paper presents a comparison between the performance predicted by a computational fluid dynamic (CFD) model and experimental measurements taken using a vortex tube produced for experimentation. Specifically, the measured exit temperatures and velocities into and out of the vortex tube are compared with the CFD model. The CFD model is a three-dimensional (3D) steady 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 investigate temperature and velocity distribution. The objective of this paper is the demonstration of the successful use of CFD in this regard, thereby providing a tool that can be used to understand the vortex mechanism clearly.

2. EXPERIMENTAL SETUP

A vortex tube was used having copper tube and standard insulators. The swirl chamber was fabricated and tangential inlet was prepared by tangentially inserting a nozzle in the chamber. Commercially available expansion valve was used. The pipe at the hot end has a diameter of 36 mm and the cold end has 28mm. The diameter of the orifice is 10 mm. The length of the total vortex tube was around 1.2 meter. Temperature at the inlet and outlet was measured by using digital thermocouple and the velocity was measured by anemometer. The experimental setup and enlarged view of several parts are shown in Fig 1 and Fig 2.

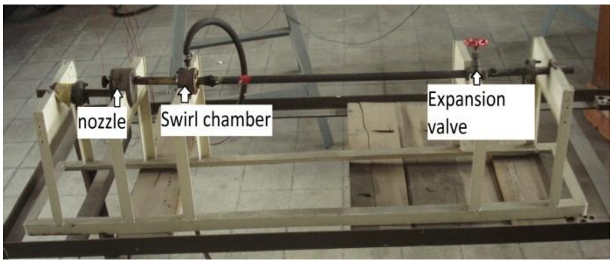


Fig 1. Experimental set up of vortex tube

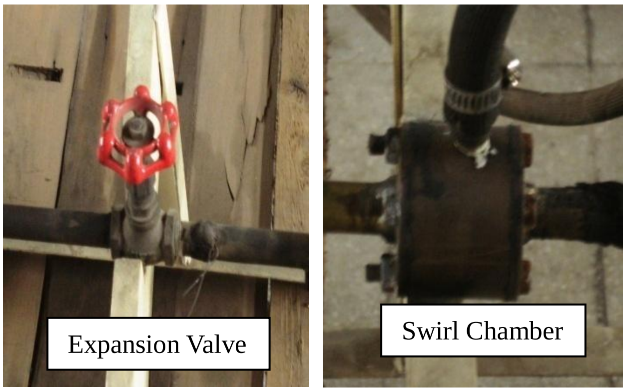


Fig 2. Different parts of vortex tube

3. EXPERIMENTAL DATA

During the experiment the data given in table 1were obtained. The inlet pressure was changed to observe the vortex characteristics. The pressure was varied between 8000- 10000 KPa. Then readings were taken after the system became steady. Four set of reading were taken.

Table 1: Experimental data

Obs. no.	Inlet Temperature (K)	Hot end		Cold end	
		Temperature (K)	Velocity (m/s)	Temperature (K)	Velocity (m/s)
1	307	328	4.2	293	6.7
2	307	330	5.6	291	7.5
3	307	329	7.4	290.5	9.6
4	307	325	10.4	290	12.5

4. CFD MODEL

The model for CFD analysis was built in different stages using different software. They are AutoCAD, Ansys ICEM-CFD, Ansys CFX. The detail of each step is given in the following subsections.

4.1 Geometry modeling

An actual model of the experimental setup was built using the dimensions of the set up in AutoCAD. All the geometric parameters were kept same except some complex parts (such as the expansion valve) were replaced with simplified geometric shapes (such as cone). The CAD model is shown in Fig 3 and Fig 4.

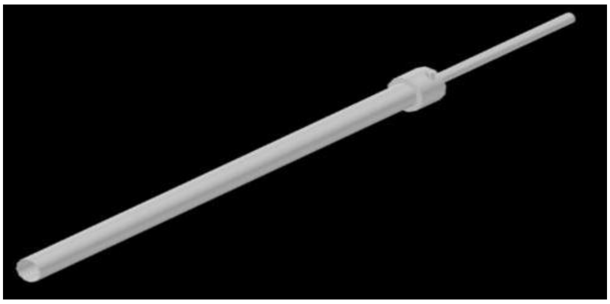


Fig 3. CAD model of vortex tube

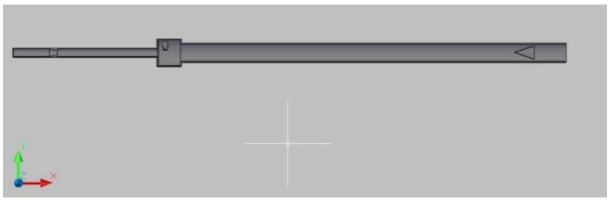


Fig 4. 2D vertical section view of CAD model

4.2 Meshing

Tetrahedral mesh elements were generated using ICEM CFD software. First the CAD model was imported in suitable file format. Then several parts of the model such as inlet, outlet, wall etc ere defined. Then mesh elements were generated of suitable size at different parts within the model. Fig 5 shows a sample of tetrahedral mesh and the model after mesh in ICEM CFD. The picture of the model after meshing is given in Fig 5.

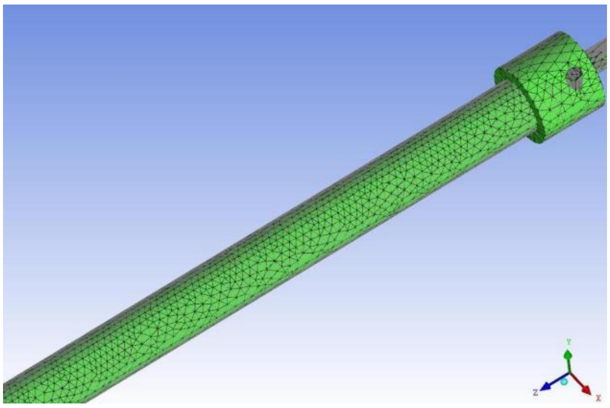
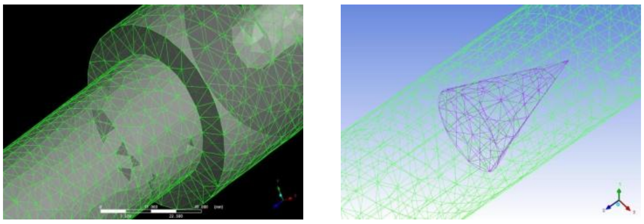


Fig 5. Model after mesh generation

The enlarged view of mesh at different parts of the model is shown in Fig 6.



a) Elements at vortex chamber b) Elements at valve

Fig 6. Mesh elements at different parts

4.3 Boundary conditions

All the boundaries were defined by boundary conditions on the model using ANSYS CFX pre software. The boundaries were inlet, two outlets (hot and cold), valve and the wall. The fluid was defined as air. The reference pressure was taken as 1 atm and the atmospheric temperature was taken as 298 K for the CFD model. All the other parameters used for the simulation are shown in tables below.

Table 2: Boundary conditions used in simulation

Parameters	**Value (initial condition)**
Inlet temperature	307 K
Outlet temperature	298 K
Outlet pressure	101.325 KPa
Wall roughness	Smooth wall
Wall heat transfer co-efficient	386 W m^{-2} K^{-1}
Wall temperature	298 K
Valve temperature	298 K
Heat transfer in valve	adiabatic

Table 3: Air properties used in simulation

Properties	**Values**
Molar Mass	28.98 Kg mol^{-1}
Specific heat capacity	1004.4 J Kg^{-1} K^{-1}
Dynamic viscosity	1.83x 10^{-5} Kg $m^{-1}s^{-1}$
Thermal conductivity	0.0261 $Wm^{-1}K^{-1}$
Thermal co-efficient of expansion	0.003 K^{-1}

4.4 Simulation

The CFD model was solved by using ANSYS CFX solver software and the results were extracted using ANSYS CFX post software. The model was solved for each of the inlet conditions. Mainly the temperature contour plot and the velocity vector diagrams were generated. The temperature contour plot helped to visualize the distribution of temperature within the vortex tube. The velocity vector diagram revealed the velocity direction and value in each section of the model. The outlet temperatures and the velocities were obtained from these diagrams. Fig 7 shows a sample of temperature contour plot and Fig 8 shows the sample of velocity vector diagram. An enlarged view of vector diagram inside the vortex chamber is also shown in Fig 9.

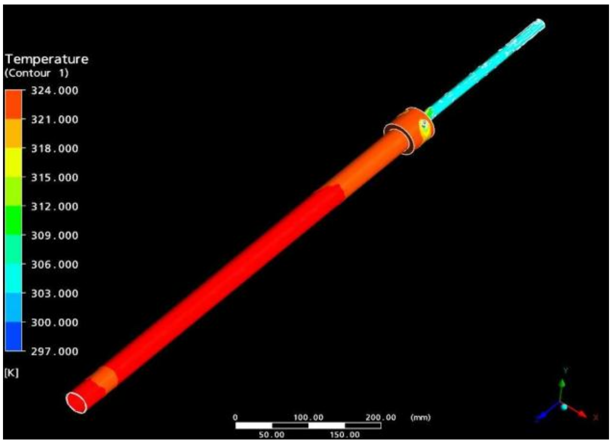


Fig 7. Temperature contour of the vortex inside the tube

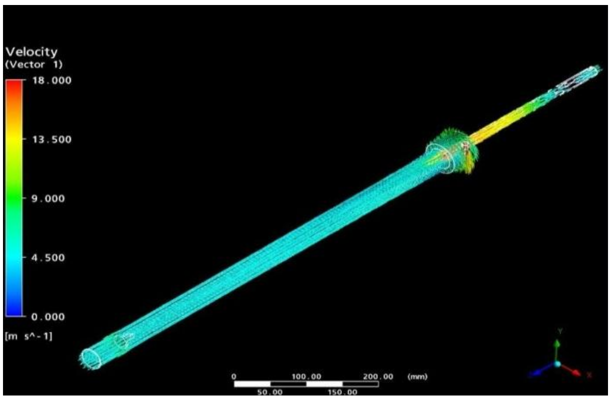


Fig 8. Velocity vector diagram of the vortex inside the tube

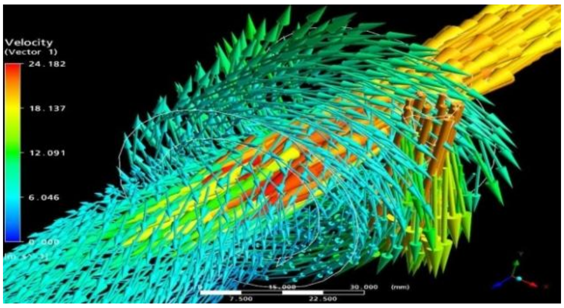


Fig 9. Velocity vector diagram of the vortex inside swirl chamber

5. COMPARISON BETWEEN CFD RESULT AND EXPERIMENTAL DATA

The comparison between experimental and analytical value is given in Table 4 and Table 5.

Table 4: Comparison of outlet velocity

Hot End		Error %	Cold End		Error%
Experimental Velocity (m/s)	Simulated velocity (m/s)		Experimental Velocity (m/s)	Simulated velocity (m/s)	
4.2	3.82	9.04	6.7	5.85	12.69
5.6	4.75	15.17	7.5	7.3	2.6
7.4	6.75	8.78	9.6	9.23	3.85
10.4	9.00	13.46	12.5	12.09	3.28

Table 5: Comparison of outlet temperature

Hot End		Error %	Cold End		Error%
Experimental Temperature (K)	Simulated Temperature (K)		Experimental Temperature (K)	Simulated Temperature (K)	
328	324.01	1.2	293	297	-1.3
330	327.97	0.6	291	306.884	-5.4
329	326.49	0.7	290.5	305.154	-5.0
325	320.67	1.3	290	296.00	-2.0

5. CONCLUSION

From the above analysis it can be concluded that the CFD model used in this study was quite effective to predict the vortex behavior in a vortex tube. Although there have been some errors in the result but these errors can be eliminated by increasing the accuracy of the model and by taking smaller time steps in simulation. This proposed CFD model of the vortex tube can be used to analyze the change of temperature and velocity within a vortex tube in a very effective way.

6. REFERENCES

1. Rudolf Hilsch, The Use of the Expansion of Gases in A Centrifugal Field as Cooling Process, *The Review of Scientific Instruments*, vol. 18(2), 108-1113, (1947). Translation of an article in *Zeit. Naturwis*. 1 (1946) 208.
2. Chengming Gao, "Experimental Study on the Ranque-Hilsch Vortex Tube", (2005) page 2
3. R.T. Balmer. "Pressure-driven Ranque-Hilsch temperature separation in liquids". Trans. ASME, J. Fluids Engineering, 110:161–164, June1988.
4. K. Stephan, S. Lin, M. Durst, F. Huang,and D. Seher, "An investigation of energy separation in a vortex tube" International Journal of Heat and Mass Transfer, Volume 26, Issue 3, March 1983, Pages 341-348.
5. M. Kurosaka, "Acoustic streaming in swirling flow and the Ranque—Hilsch (vortex-tube) effect", Journal of Fluid Mechanics (1982), 124: 139-172, Copyright © 1982 Cambridge University Press.
6. Merwin Sibulkin, "Unsteady, viscous, circular flow Part 3. Application to the Ranque-Hilsch vortex tube", Journal of Fluid Mechanics (1962), 12: 269-293, Copyright © 1962 Cambridge University Press.
7. T. Passot, H. Politano, P.L. Sulem, J.R. Angilella and M. Meneguzzi, "Instability of strained vortex layers and vortex tube formation in homogeneous turbulence", Journal of Fluid Mechanics (1995), 282: 313-338, Copyright © 1995 Cambridge University Press.
8. Upendra Behera, P.J. Paul, S. Kasthurirengan, R. Karunanithi, S.N. Ram, K. Dinesh and S. Jaco, "CFD analysis and experimental investigations towards optimizing the parameters of Ranque–Hilsch vortex tube", International Journal of Heat and Mass Transfer, Volume 48, Issue 10, May 2005, Pages 1961-1973.
9. N. F. Aljuwayhel, G. F. Nellis, "Parametric and internal study of the vortex tube using a CFD model", International Journal of Refrigeration, Volume 28, Issue 3, May 2005, Pages 442-450.
10. H.M. Skye, G.F. Nellis, "Comparison of CFD analysis to empirical data in a commercial vortex tube", International Journal of Refrigeration, Volume 29, Issue 1, January 2006, Pages 71-80.
11. M. H. Saidi, M. S. Valipour, "Experimental modeling of vortex tube refrigerator", Applied Thermal Engineering, Volume 23, Issue 15, October 2003, Pages 1971-1980.