A Review of Vortex Flow Field and Temperature Separation of a Ranque–Hilsch Vortex Tube

Chaitanya Kotecha[1], Abhishek Singh[2]
[1]PG Scholar, [2]Assistant Professor
[12]Department of Mechanical Engineering, Shri Shankaracharya Technical Campus, SSGI, Junwani, Bhilai, Chhattisgarh, (C.G.)

Abstract

The vortex tube is a quiet and simple device used commercially and very efficiently to develop cold and hot air separately from compressed air at the single supply and different temperatures. When compressed air enters the inlet nozzle, vortex flows are produced in it. In which the inlet nozzle plays an important role for this. When the air enters the inlet nozzle and forms the vortex, the hot and cold air inside the vortex tube begins to separate and flow out of the different exit zones. For this reason, it is important to be efficient on the basis of use in the air which is getting exit, which is very important to research.

Keywords*:* *Vortex Tube, Cold Inlet, Hot Inlet, CFD Analysis.*

1. Introduction

The vortex tube (VT) is a device that generates varying flows of hot and cold gases from one source of compressed gas. The vortex tube was invented quite by accident in 1933 by George Ranque and later developed by Hilsch (1947). In memory of their contribution the Vortex tube is also known as Ranque-Hilsch Vortex Tube (RHVT). It contains the parts: inlet nozzle, vortex chamber, cold-end orifice, hot-end control valve and tube. The working principle of the VT is as shown in Figure 1.1. The compressed fluid is introduced tangentially into the vortex tube through the tube, due to the cylindrical structure of the tube and, depending on its pressure and inlet velocity, at high speeds the vortex conducts a circular motion within the tube. The pressure difference between the tube walls is less than the velocity at the center of the tube due to the effects of wall friction.

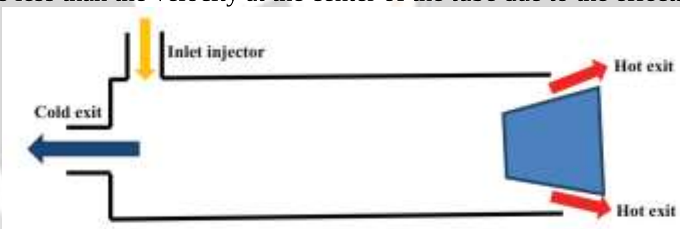

Figure 1.1 Common air separator of RHVT

2. Literature Review

Sky et al. (2006) investigated and created a CFD model of a commercial VT to be used as a design tool to optimize the performance of a VT. The model was developed using a symmetric stationary axis two-dimensional model (with vortex) that used the standard turbulence equation k-epsilon. The K-Epsilon RNG turbulence model was investigated; However, it was found that the difference between measured and predicted energy separation increases. Furthermore, they concluded that in this context, the model is useful as a cost-effective tool and saves time for designing VT.

Yadav et al., (2013) studied that concentrated research has been carried out over the years, the efficiency is still low. So, in order to improve the performance of the vortex tube, an innovative modification is applied in the present work i.e., to make the forced vortex at the cold end to strike back again towards opposite end, forming one more forced vortex flow. Thus, the modified VT was named as dual forced flow vortex tube (DFFVT), was investigated experimentally to study the energy separation. The effect of pressure and end plug sizes at both ends on temperature drops are investigated and presented in his work.

Gupta et al., (2014) studied impacts of affecting factors on the formation are analyzed herein. An intense free vortex and a surface depression formed above the drain orifice. It consists an air core region surrounded by free vortex. Free vortex surface extended to outlet drain orifice. In result analysis it is found out that Free surface vortex adjacent to Inner air core region was height dependent, vortex radius expanded as height increased and on comparing this radius- height ratio for different outlet areas, it was established that with increment in outlet area, higher values of vortex radius obtained for same values of height though profile pattern was invariable.

Mihalić et al., (2014) presented a method of numerical volume control with an unstable solver and it was demonstrated that the des turbulence model is a valuable tool for flow analysis in a centrifugal pump. Given that the DES turbulence model takes a much shorter computation time than the LES turbulence model, this is a very useful fact that arose from its research. At low flow rates, the contribution of the vortex rotor to the head was significant, while not consuming additional driving energy. This showed that a part of the energy that would be lost in centrifugal pumps is recovered by coherent structures.

Aswalekar et al., (2014) presented the numerical investigation of the vortex tube intends to bring out unexplored features. The work was based on vortex tube of L/d ratio 10 having two straight nozzles of circular cross-section. Solid Works 2010 software is used to generate the 3-D model of cavity domain. The flow is analyzed using ANSYS FLUENT 13.0 wherein the, pressure distribution and temperature distribution laterally the length of VT was studied. The path lines are observed to match with the theoretically stated generally for vortex tube.

Vlček (2014) created a model for simulating the formation of vortex on the free water surface in a stirred vessel without baffles. The sensitivity analysis showed that the resulting flow didn't change much with coarseness of mesh. But coarseness of mesh has effect on the sharpness of the phase interface, therefore, it is appropriate to use finer grid at the place of expected interface. The best turbulent model used for stationary simulation of stirred vessel without baffles was Reynolds Stress - Quadratic Pressure Strain. A good water surface shape was obtained with this model compared with the experiment. Prediction of the depth of vortex by simulation was about a centimetre smaller than that was found in the experiment.

Vhankade (2015) studied the phenomenon of temperature distribution in confined steady rotating gas flows. The simple counter-flow vortex tube consists of a long hollow cylinder with tangential nozzle at one end for injecting compressed air. The flow inside the vortex tube can be described as rotating air, which moves in a spring shaped vortex track. The peripheral flow moves toward the hot end where a hot end plug was placed and the axial flow, which was forced back by the plug, moves in the opposite direction towards the cold end.

Karthik (2015) designed and computed of COP of vertex tube. The calculations for flow rates are measured and the designs are similar to what was taken by Hilsch, Reynold and Albohrn.

Kim et al., (2015) conducted an experiment on a dedicated cesspool model installed in the KMOU laboratory and numerical simulations were performed in ANSYS CFX 13.0. Using both of these methods, the vortex and vortex angles were predicted from the results obtained with and without AVD and it was found that the difference in the results may be due to some unavoidable errors in experiments, such as water flow due to which The vortex meter fluctuates and does not spin completely, and the congenital problem that CFDs cannot actually replicate experimental conditions. Despite this, CFD results closely follow the trend of experimental results.

Yadav 2015 studied with the modified vortex tube (DFFVT), the secondary flow from the stagnation point is made to leave by afford a way through the centre of conical valve at the hot end which results in development of one more forced vortex flow. Also, to improve the process a conical valve is provided at the cold end (towards injection end). Thus, the multi circulation because of secondary flow can be avoided and preferred swirling flow can be improvised. This results in further increase of temperature drop.

Thakare et al., (2015) obtained results during the study with experimental results. The observation of axial velocity provides an idea about the inverse of the flow. The 2-D axial model can be successfully used to predict temperature separation and flow behavior within the VT. In contrast to the maximum separation of cold end temperatures, a maximum separation of cooling power is achieved for a fraction of the cold mass. The structure of the passage lines clearly identifies the area adjacent to the cold exit, as well as the presence of secondary vortices within the tube.

3. Conclusions

Present working principle of vortex tube, temperature (energy) separation phenomenon and geometrical parameters affecting the performance and CFD analysis of vortex tube. This work also includes governing equations and boundary conditions for vortex tube analysis. Hypotheses of temperature separation are pressure gradient, viscosity, turbulence, temperature gradient and secondary circulation. Furthermore, work shows that different types of nozzle profiles/shape, number of nozzle and axial nozzles angle are evaluated by CFD analysis. Boundary conditions were modified to obtain the required vortex flow. Due to the complex structure of the internal flow, the reason for generating hot and cold flows from a single injection into a vertex tube have not been identified and the flow behaviour inside the vertex tube, therefore, remains unclear. For the purpose of concluding a reasonable explanation for the temperature separation in the vortex tube, this work has presented a CFD analysis focusing on the flow behavior inside the vortex tube.

References

[1] H. M. Skye, G. F. Nellis, and S. A. Klein, "Comparison of CFD analysis to empirical data in a commercial vortex tube," *Int. J. Refrig.*, vol. 29, pp. 71–80, 2006.

[2] G. M. P. Yadav, P. M. Reddy, and B. U. M. Gowd, "Experimental Investigation on Temperature Separation of Dual Forced Flow Vortex Tube," *Int. J. Eng. Res. Technol.*, vol. 2, no. 6, pp. 1629–1634, 2013.

[3] S. Gupta, J. P. Panda, and N. Nandi, "A model study of free vortex flow," in *International Conference on Theoretical, Applied, Computational and Experimental Mechanics*, 2014, vol. 12, pp. 1–9.

[4] T. Mihalić, Z. Guzović, and A. Predin, "CFD flow analysis in the centrifugal vortex pump," *Int. J. Numer. Methods Heat Fluid Flow*, vol. 24, no. 3, pp. 545–562, 2014.

[5] U. V Aswalekar, R. S. Solanki, V. S. Kaul, S. S. Borkar, and S. R. Kambale, "Study and Analysis of Vortex Tube," *Int. J. Eng. Sci. Invent.*, vol. 3, no. 11, pp. 51–55, 2014.

[6] I. P. Vlček, "Steady CFD simulation of central vortex formation at the free surface in the vessel without baffles stirred by impeller with three curved blades," in *17th Conference on Process Integration, Modelling and Optimisation for Energy Saving and Pollution Reduction*, 2014, vol. 2, pp. 1–14.

[7] P. B. Vhankade, "Design and Manufacturing of Vortex Tube," *Int. J. Sci. Res.*, vol. 6, no. 4, pp. 263–266, 2015.

[8] S. Karthik, "Design and Computation of COP of Vortex Tube," *Int. J. Sci. Eng. Res.*, vol. 6, no. 4, pp. 434–438, 2015.

[9] C. G. Kim, B. H. Kim, B. H. Bang, and Y. H. Lee, "Experimental and CFD analysis for prediction of vortex and swirl angle in the pump sump station model," in *International Symposium of Cavitation and Multiphase Flow*, 2015, vol. 72, pp. 1–7.

[10] G. M. P. Yadav, "CFD Analysis of Temperature Separation in Modified Vortex Tube with Dual Forced Vortex Flow," *Eur. Int. J. Sci. Technol.*, vol. 4, no. 8, pp. 47–60, 2015.

[11] H. R. Thakare, A. Monde, B. S. Patil, and A. D. Parekh, "Numerical Investigation of Flow Characteristics in Counter Flow Vortex Tube," *Procedia Eng.*, vol. 127, pp. 170–176, 2015.