Available online: www.ijsrmes.com

International Journal for Scientific Research in Modern Engineering and Science

Published in: Vol. 1, Issue 2, pp. 8-13, May-June 2026

ANALYSIS OF VORTEX FLOW FIELD AND TEMPERATURE SEPARATION IN A RANQUEHILSCH VORTEX TUBE WITH A MODIFIED NOZZLE

Chaitanya Kotecha[1]*, Abhishek Singh[2]

[12]*Department of Mechanical Engineering, SSTC-SSGI, Bhilai (C.G.), INDIA.*

Abstract

The vortex tube is a mechanical device without moving parts accomplished of generating hot and cold air flow from compressed air. In this study show that the use of a vortex tube improves the cooling performance of the vortex tube. A numerical investigation has been carried out to study the effect of the use of vortex tubes and to find the temperature separation of hot and cold air. The results have been predicted by using computational fluid dynamic software which the energy separation effect inside the tube also has been created using the standard model RNG turbulence model. Firstly, the work has been done with the adjacent test of the grid to get good and accurate simulation results. A comparison of the present results and the work of available literature has been validated and a good quantitative and reliable agreement has been found between the simulation results..

Keywords: *Vortex tube; Colt temperature separation; Hot temperature separation; Ansys; CFD.*

* *Corresponding author*

1. Introduction

The vortex tube (VT) is a device that separates a single stream of compressed gas into distinct flows of hot and cold air. The vortex tube was accidentally invented in 1933 by George Ranque and subsequently refined by Hilsch in 1947. In honor of their contribution, the Vortex tube is also referred to as the Ranque-Hilsch Vortex Tube (RHVT). It includes the following components: inlet nozzle, vortex chamber, cold-end orifice, hot-end control valve, and tube. The VT operates according to the principle illustrated in Figure 1.1. The compressed fluid enters the vortex tube at an angle, thanks to the tube's cylindrical shape; depending on its pressure and inlet speed, it moves rapidly in a circular path inside the tube. The pressure difference across the tube walls is lower than the velocity at the tube's center, caused by wall friction. Consequently, energy moves from the fluid in the central region to the fluid along the tube wall. After reaching a pause point, the cold fluid exits the moving tube in the opposite direction of the main flow, while the warm fluid exits in the same direction as the main flow. RHVT is commonly employed for both cooling and heating purposes.

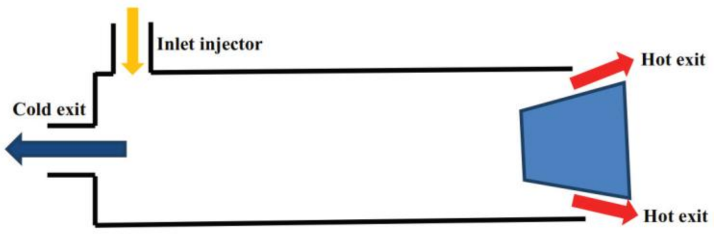

Figure 1. Common air separator of RHVT

2. Problem Statement

Mostly, for simplify the process and calculation time, the 2-D vortex model were used by various literature. Due to 2-D model, most of parameters and equations is neglected, therefore, overcome this issue, 3-D model analysis has been done, which are provides the appropriate result. In present work the 3-D model and analysis has been done. For geometric modeling, meshes and simulations of models have been performed by Creo 3.0 and Ansys Fluent 16.2 software, respectively.

3. Methodology

In this chapter, the methodology of the present work is presented. The government equations used for analysis are the continuity equation, the momentum equation and the energy equation along with the state equation of an ideal gas and these equations are solved by the CFD Ansys fluent software. The equation of motion of vortex tube has been solved using FEV tool (ANSYS- Fluent) as the equation of motion for a vortex tube is difficult to visualize therefore some FEV tool is the only solution method for analyzing thermo physical characteristics of vortex tube.

Figure 2. 3-D model of a vortex tube with inlet and exit conditions

4. Results & Discussions

In this chapter, detailed discussions about the thermal effect and flow effect field characteristics of vortex tube with different nozzle inlet shape, number of nozzles, and axial angle have been presented. For 3-D space, a 3-D model and for viscous models a k-ε (2 equa.), simulation has been required at a steady-state time-dependency with RNG turbulence model with a standard-wall-function condition has been considered for present models.

4.1 Grid Independence Study

In present work, the grid independence study has been carried out with varying the size of meshing or number of elements. After grid impendency test it has been found that the beyond the 58648 elements, results are nearly or same. The grid impendency test is provide the less effort and time to achieved the results..

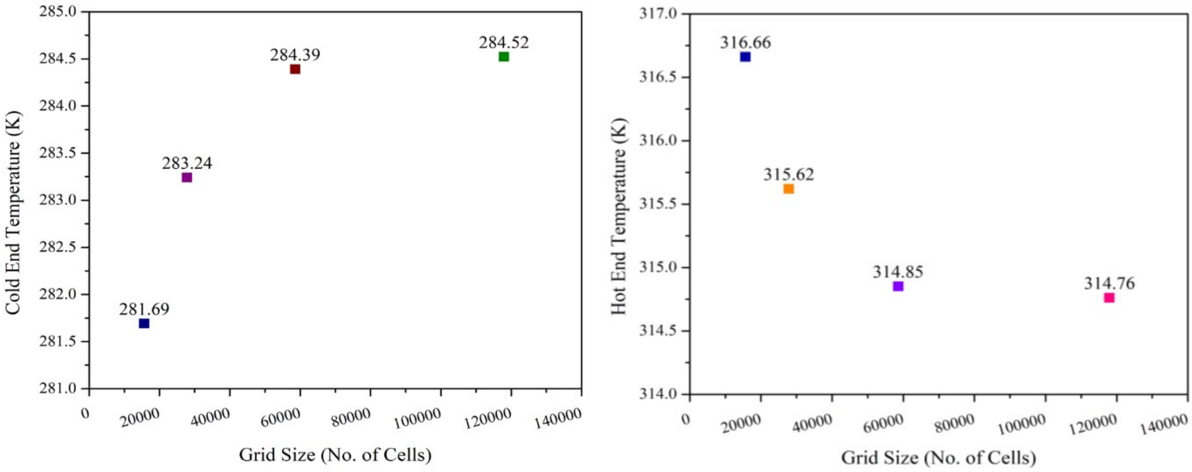

Figure 3. Grid independence test on cold and hot end temperature

4.2 Validation Numerical Analysis

The current CFD results as shown in Figure 4 are same or nearly with the experimentally and CFD results available from Sky et al. [2], Farooq and Farooq [3] and Thackeray et al. [15] for recognition. Figure 4 It is clearly demonstrated that the results of the present or proposed CFD work are in good agreement with the results available from Sky et al. [2] That is the CFD result of Skye et al. [2] For the separation of cold and hot temperatures (cold and hot temperature differences) and reliable with Farooq and Farooq's CFD results [3] and Thakar et al [15] this implies that the simulation method used in this work Despite the use of the RNG turbulence model, Wajib and Farooq [3] and Thackeray et al [15] are reasonable.

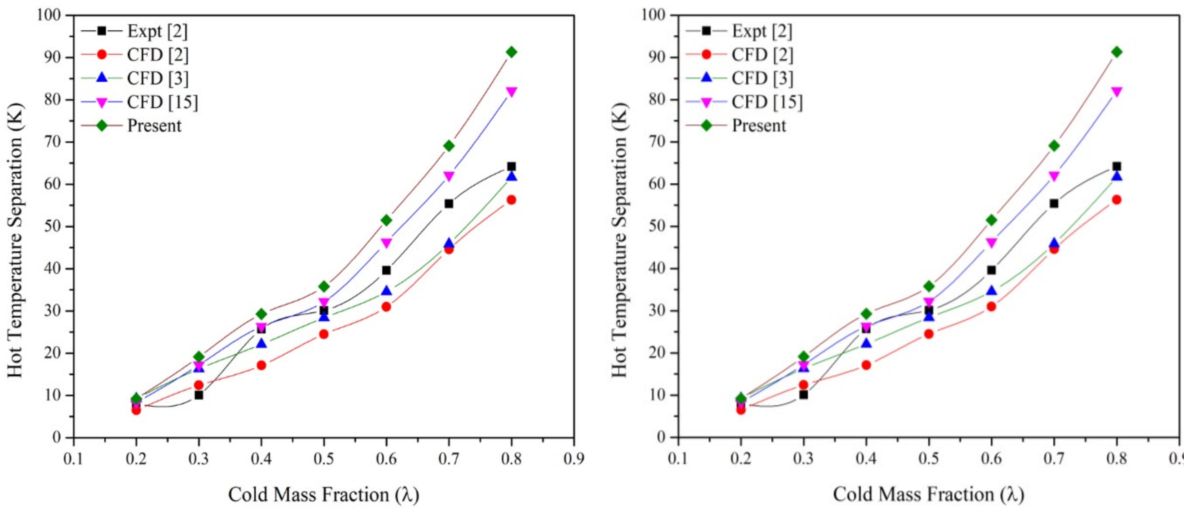

Figure 4. Comparison of experimental [2] and CFD [3] [15] results from present hot temperature separation

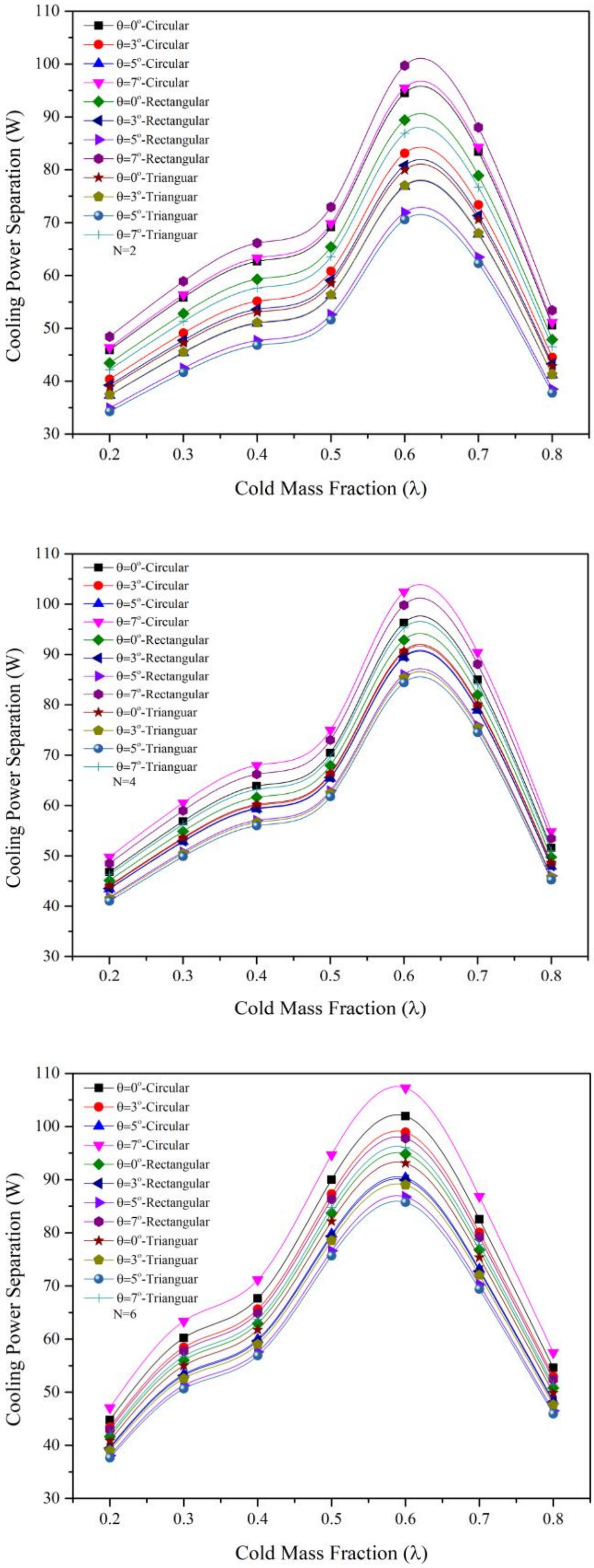

Figure 5. Variation of cooling power separation (cooling capacity) over different shaped and angle inlet nozzle for 6 inlet nozzles (N=2, 4 and 6)

Figure 5 depicted that the cold temperature vs. cold temperature fraction in the vortex tube is represented to analyze the effect of the magnitude of the axial angle on the overall performance of the vortex tube.

5. Conclusions

In this study, the temperature difference or separation occurring within the vortex tube has been investigated using CFD, which serves as a predictive tool. The model was developed in Creo, followed by a meshing analysis of the three-dimensional model. Several factorsincluding the number of inlet nozzles, their shape, and axial anglecan influence the vortex tube's performance. First, the grid impedance test has been conducted to ensure accurate and reliable simulation outcomes. Then, the work was validated and compared with existing literature, i.e., Skype et al. [2], Farouk and Farouk [3], and Thakare et al. [and found strong quantitative and consistent alignment across the simulation results. Additionally, the design has been refined to enhance the thermal performance of vortex tubes.

References

[1] V. Singh, S. Srivastava, R. Chaval, V. Vitankar, B. Basu, and M. C. Agrawal, "Simulation of gas-solid flow and design modifications of cement plant cyclones," in *Fifth International Conference on CFD in the Process Industries*, 2006, vol. 13, pp. 1–8.

[2] 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.

[3] T. Farouk and B. Farouk, "Large eddy simulations of the flow field and temperature separation in the Ranque–Hilsch vortex tube", *Int. J. of Heat and Mass Trans.*, vol. 50, issue 23, pp. 4724–4735, 2007.

[4] N. Ozalp and D. Jayakrishna, "CFD analysis on the influence of helical carving in a vortex flow solar reactor," *Int. J. Hydrogen Energy*, vol. 35, no. 12, pp. 6248–6260, 2010.

[5] H. Pouraria and M. R. Zangooee, "Numerical investigation of vortex tube refrigerator with a divergent hot tube," in *Energy Procedia*, 2012, vol. 14, pp. 1554–1559.

[6] 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.

[7] 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.

[8] 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.

[9] 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.

[10] 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.

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

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

[13] 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.

[14] 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.

[15] 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.