Available online: www.ijsrmes.com

International Journal for Scientific Research in Modern Engineering and Science

International Journal for Scientific Research in Modern Engineering and Science, 1(1): 38-43 (2020)

VORTEX FLOW FIELD AND TEMPERATURE SEPARATION ANALYSIS OF A RANQUE–HILSCH VORTEX TUBE WITH 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 produces 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 speed at the center of the tube due to the effect of wall friction. As a result, the fluid in the central region transfers energy to the fluid in the tube wall. The cold fluid leaves the moving tube against the direction of the main flow after a pause point, while the warm fluid leaves the tube in the main direction. RHVT is widely used for cooling and heat

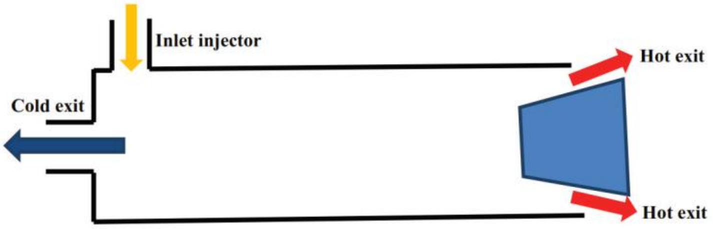

Figure 1. Common air separator of RHVT

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..

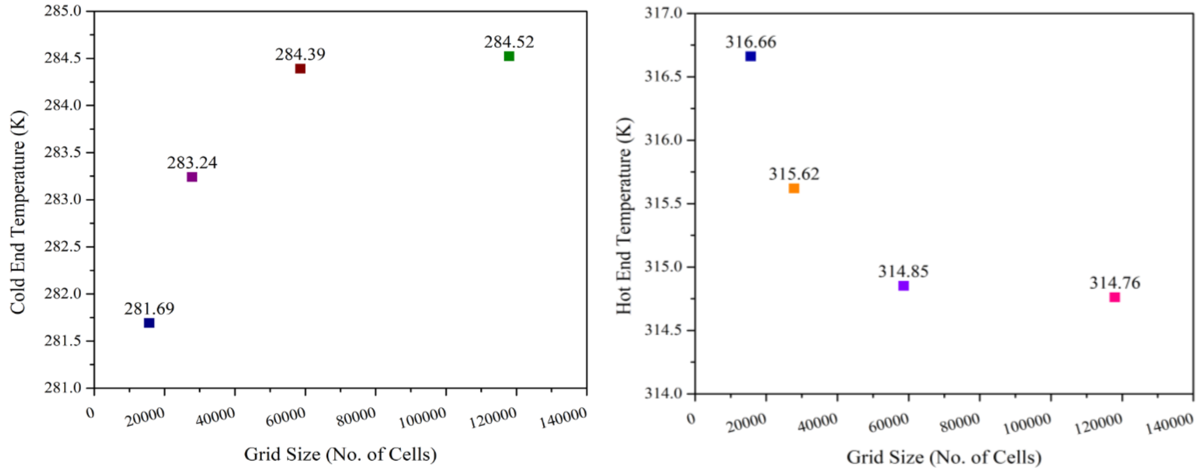

Figure 3. Grid independence test on cold and hot end temperature

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.

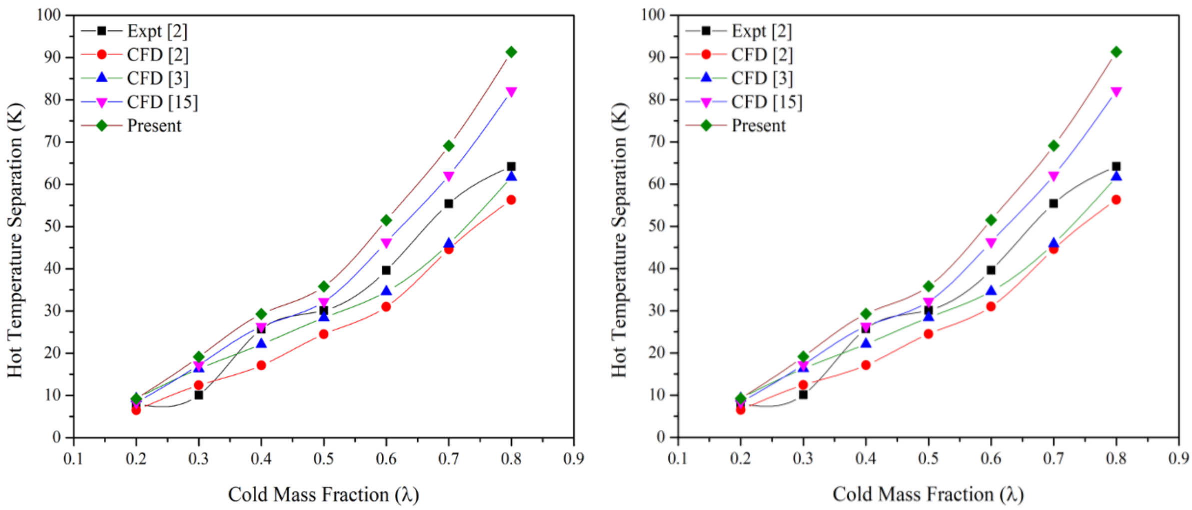

Figure 4. Comparison of experimental [2] and CFD [3] [15] results from present hot temperature separation

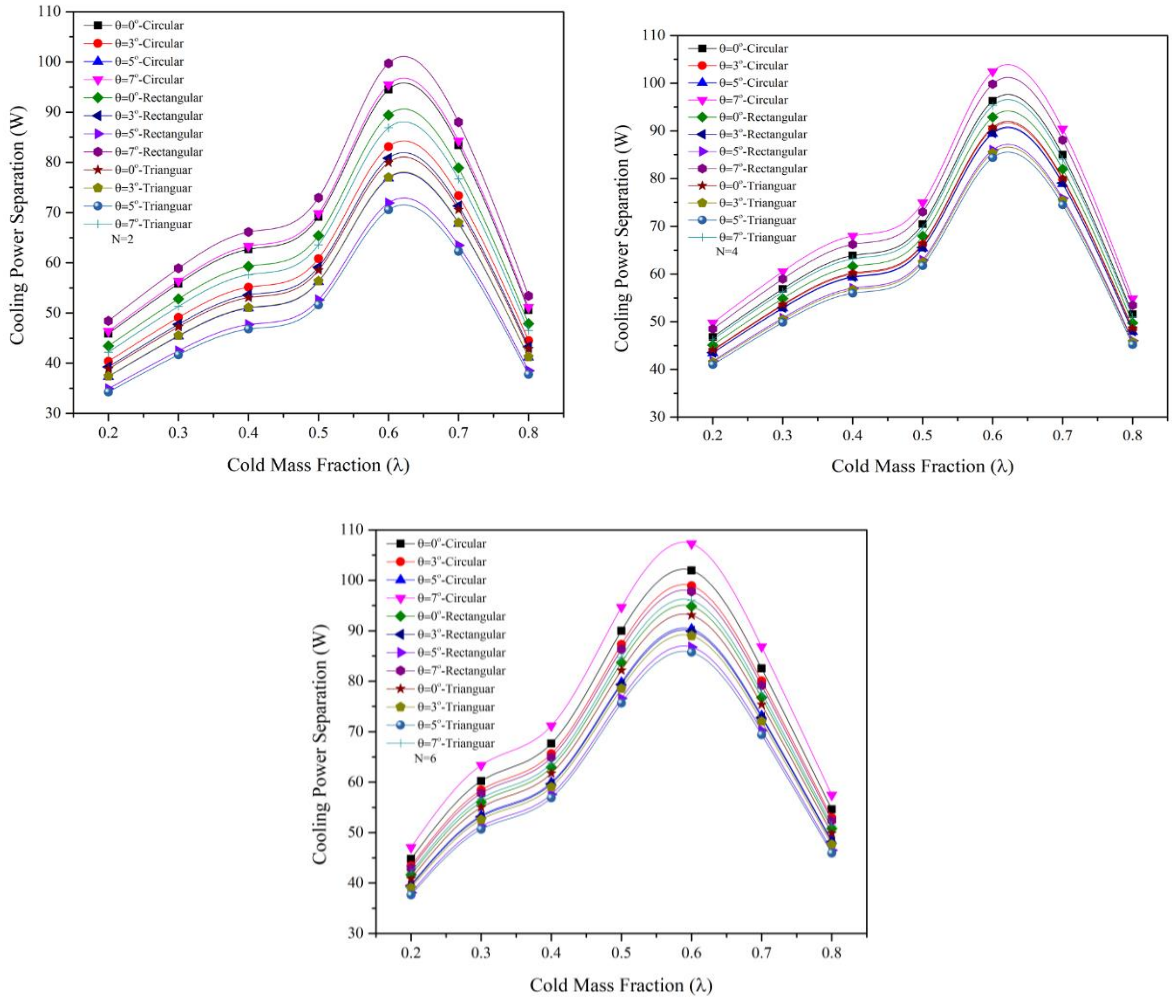

Figure 5. Variation of cooling power separation (cooling capacity) over different shaped and angle inlet nozzle for 6 inlet nozzles (N=2, 4 and 6)

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 present work, the temperature difference or separation show inside the vortex tube has been studied with the help of CFD (as a predictive tool). The model has been created in Creo and then the study of meshing of a three-dimensional model has been done. Several parameters such as number of inlet nozzles, shape and axial angle, which can affect the performance of the vortex tube. Firstly, the work has been done the grid impendency test for good and near to accurate simulation results. Then, the work has been validated and compared the present results from available literature i.e. Skye et al. [2], Farouk and Farouk [3] and Thakare et al. [15] and found good quantitative and reliable agreement among the results from the simulation. Also, the work has been optimized design for improve the thermal characteristics 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.