Open Journal of Fluid Dynamics, 2020, 10, 151-163
https://www.scirp.org/journal/ojfd
ISSN Online: 2165-3860
ISSN Print: 2165-3852

Simulation and Experimental Analysis of Vortex Tube Using Steel Material

Muhammad Bilal[1,2], Yaolong Chen[1,2*], Jun Zha[1,2], Naveed Ullah[1,2]

[1]School of Mechanical Engineering, Xi'an Jiaotong University, Xi'an, China
[2]State Key Laboratory for Manufacturing System Engineering, Xi'an Jiaotong University, Xi'an, China
Email: Bilalktk@stu.xjtu.edu.cn, *chenwei@mail.xjtu.edu.cn, jun_zha@xjtu.edu.cn, naveed01@stu.xjtu.edu.cn

How to cite this paper: Bilal, M., Chen, Y.L., Zha, J. and Ullah, N. (2020) Simulation and Experimental Analysis of Vortex Tube Using Steel Material. *Open Journal of Fluid Dynamics*, **10**, 151-163.
https://doi.org/10.4236/ojfd.2020.103010

Received: June 2, 2020
Accepted: June 28, 2020
Published: July 1, 2020

Abstract

Experimental and computational fluid dynamics (CFD) are investigated through the vortex tube system. The benefit of vortex tube is a counter flow type tube, which has further designed and fabricated for investigation. The whole set up is consisting of a simple device that can separate a single stream of compressed air into two streams; one is at high temperature and the other is lower temperature following an inlet gas stream. The advantages of this tube are their compactness, safety, and low equipment cost mainly used in cooling and heating applications. This study addressed three-dimensional flows; the domain is using computational fluid dynamics (CFD) method and experimental approach to optimize the direction of RHVT. Through the CFD analysis, the best cold end diameter (dc), number of nozzles, and the best parameters for obtaining the highest hot gas temperature and lowest cold gas temperature were obtained and verified by experimental procedures.

Keywords

Ranque-Hilsch Vortex Tube, Design Analysis & Fabrication, Experimental of Vortex Tube, Steel Material

1. Introduction

A vortex tube is a thermal device that can produce hot and cold streams continually by using only compressed gas. Air is commonly used as a working medium in the vortex tube; hence, the vortex cooling is an environmentally friendly system. The most important advantage of vortex is fast cooling in a short duration and can be adjustable over a wide temperature range and endure harsh environments, having no moving parts thus with little requirement for maintenance. The device is also called Ranque-Hilsch tubes and Ranque-Hilsch vortex tubes, named after the people who discovered a tube was a French physics student named Georges Ranque in the year of 1930; several studies presented the principle of finding on the unique energy separation phenomena in the vortex tube. Compressed gas at a higher pressure than atmospheric enters the tube inlet tangentially passing through radial nozzles into the swirl or vortex generation chamber where it creates a high-speed flow with thousands of rotations per second. Two different air-streams produced then flow along the tube where a high temperature stream is found at the outer region while a low temperature stream is showed at the central region as shown in Figure 1(A). The cold and the hot streams have the main tube at the opposite end for the counter-current flow type but leave on the same side in different end for the con-current flow type. The first type is widely used accordingly; these configurations were studied in the current research as seen in Figure 1(A). Vortex tube is composed of inlet nozzles, diaphragm, cold orifice, hot control valve, hot tube, and chamber the schematic diagram of a vortex tube as given in the Figure 1(B).

Compressed air is passing to the vortex tube through the nozzle (a) in the nozzle the air injected with high speed velocity and enters into the chamber. Tangentially it converts it into a vortex. Form (c) this vortex is flow is created in the chamber and air travels in spiral like motion along the periphery of the hot side. This flow is restricted by the valve. Inside (d) the hot tube, both inner stream of air and outer stream of air rotate with same angular velocity. The vortex travels along the tube (e) tube through the diaphragm. (f) Part of this air flows returned closer to the diaphragm (b). It then leaves the tube via the diaphragm (c); then air passes through the valve. By using adjusting the valve beginning the quantity of cold air and the temperature drop may be numerous. The maximum drop is obtained for a particular establishing of the valve. A decrease

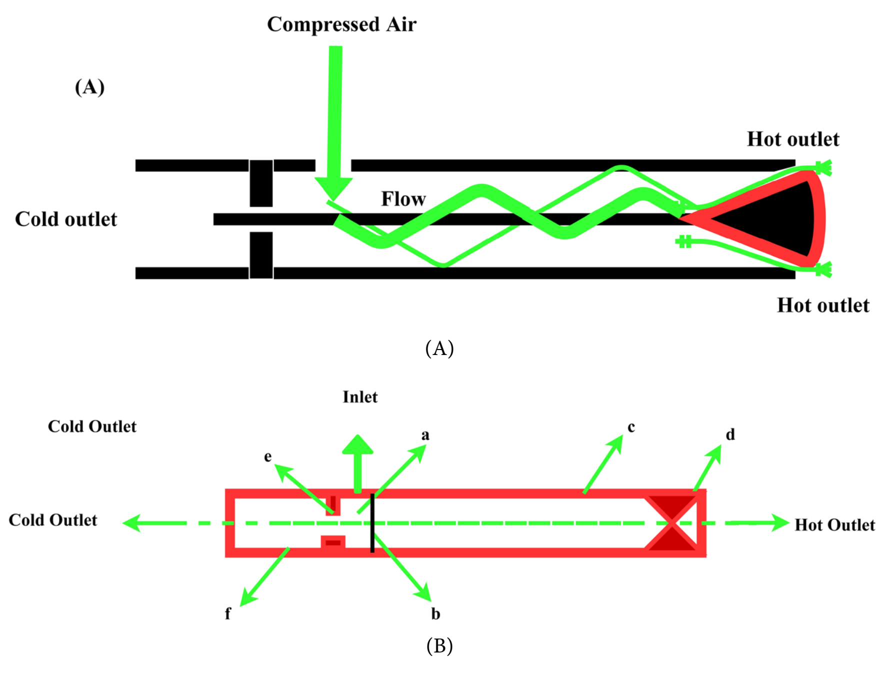

Figure 1. (A) (B) Layout of vortex tube.

in temperature drop will end result by way of lowering the valve establishing underneath this opening. Due to the vortex tube's energy separations computation of the complex flow phenomena is a challenging task. Several theories were formulated to explain the phenomena of temperature separation in the vortex tube. Many studies have been conducted over the strength of separation phenomenon, the effects and performance of the vortex tube, consequences of geometrical parameters on the cold and hot temperatures which are lately curved vortex tube turned into popular interest for the researchers. Currently, computational fluid dynamics (CFD) modeling has been successfully utilized to explain the basic principles behind energy separation. CFD analysis helped to recognize the energy separation and convey analysis phenomenon as much as a certain extent. Therefore, a model of the vortex has been designed and fabricated to examine the set-up which can show the effect of geometrical parameters on the overall performance of vortex tube [1] proposed that the switch of Tangential shear work from inner to outer fluid layers is the cause of energy separation. Ranque [2] proposed that the work switch as a result of compression and enlargement results is the primary reason for the temperature separation within the tube. Harnett and Eckert [3] present the phenomena occurring due to turbulent rotating flow with solid body rotation inside the vortex tube during the temperature-energy distribution. Because turbulent movement has a huge spectrum of eddy sizes, and big swirl (related to low-frequency fluctuations) and small swirl (associated with excessive frequency fluctuations) can coexist inside the equal extent of fluid, they recommend that the wide spectrum of vortex is the cause for the phenomena of temperature separation. Ahlborn and Groves [4] proposed that the embedded secondary flow is the phenomena that purpose the energy separation. They measured the axial and azimuthal speed components to attain the conclusion. Stephan [5] proposed that the formation of vortex on the inside wall of the vortex tube is the purpose that drives the fluid motion inside the vortex tube. Good efforts are flow-smart secondary flows that appear in a boundary layer go with the flow along a concave wall. If the boundary layer thickness is corresponding to the radius of curvature, the centrifugal action creates a pressure variation across the boundary layer. This leads to the centrifugal instability of the boundary layer and the consequent formation of better efforts [6] [7]. There work consists of effects of change in lengths and diameters of warm and cold tubes form of entrance nozzle advanced three-dimensional numerical version to apprehend the go with the flow traits and energy separation phenomenon [8].

Using a 3-D CFD model has very small millions of cells to analyze the flow field of RHVT. Authors reported that drag force caused due to the pressure difference between the flow field in RHVT and cold end acts constantly on particles that moving towards the hot end. When there is no particle left with any momentum to flow opposite of this pressure gradient, however, the axial velocity close down to zero and later on inverse its direction of flow, by moving towards the cold end [9]. The main reason of difficulties in obtaining experimental measurements within the vortex tube, there has no longer been consistent information to do with the flow behavior, by using of a special pitot tube and thermocouple mechanism to measure the pressure, velocity and temperature separation inside a counter flow vortex tube with Nitrogen. It shows that the cone angle has an influential effect on the performance of the vortex tube [10]. The Comparing of CFD model with the experimental measurement uses different diameter D, 2 D, 0.5 D, 0.25 D, 0.125 D from that they conclude that for a small diameter has a low effect [11]. Behavior on energy separation and flow field properties of a vortex tube by utilize helical nozzles. Simulated and analyzed numerically the flow and thermal field behavior of parallel Flow vortex tube [12]. Secondary zone model is found at the core region near the inlet to the middle of the vortex tube [13]. In recent years, the mechanism of (CFD) computational fluid dynamics modeling has been advanced for more surveys and simplification. Some researchers utilized a CFD model analysis to study the flow field and the temperature separation phenomena and conclude that separation occurs when an inlet pressure is sufficiently high [14] [15] [16] [17]. Some previous studies had shown that the CFD model tasted the velocity, pressure, and temperature inside a vortex tube is basic factors for the separation of hot and cold temperatures [18] [19] [20].

Conduct a CFX two-dimensional code to simulate the separation phenomenon in VT with gaseous air at cryogenic temperature as a working fluid. The results show that the energy separation decreases with an increase in hot outlet mass fraction [21] [22] [23]. The effect of varying the cone valve diameter and also increasing the inlet air pressure on the performance of the inflow vortex tube using two nozzles diameter investigated experimentally [24]. The present study provides a comparison of the results obtained by using these turbulence models [25].

The revised literature noticed that the quality characteristic relating to the effect like temperature difference, increased with the increase of inlet pressure and cold mass fraction and decreased with the increase in nozzle number [26] [27] [28] [29] [30]. The main objective of the current research is to determine the optimum value of cold and hot end temperature and mass of the cold air with improving the COP and cooling effect by designing the vortex tube with the best possible design parameters with the help of CFD and experimental techniques. From the past research, experimental results provide comparisons with theoretical approximations and thus validation of the suggested method has been described.

The first part of the current study focuses on the experimentation of the vortex tube using CFD by preparing the three-dimensional flow domains to find out the optimum value of cold and hot end temperature. The second part includes the design and manufacturing of diverge model of vortex tube which is given in **Figure 2**. The experimental test set up is carried out in a laboratory and an experiment was performed to determine the optimum value of cold and hot end temperature.

Figure 2. Fabricated model of vortex tube.

In the last part of this work consists of, comparison of the results obtained from CFD analysis and experimental studies.

Hence, on the base of the geometrical parameter is given in **Table 1**, A fabricated model was obtained as shown in **Figure 2**.

2. Materials and Methods

The principal objective of the current research is to find out the optimum value show off the results of the CFD and experimental modeling of the vortex tube and evaluate with each different to get fine result. Experimenting with steeliness metal material, converting all geometric parameters of the vortex tube are few fundamental newly sampled changes shown in **Figure 2**.

A model of vortex tube is created in solid work by using basic parameters and after that assembly is imported into workbench analysis for the uses of fluent flow. The vortex tube boundary conditions and dimensions are used.

This study is similar to those used by Behera [25]. In the three-dimensional CFD model of the vortex tube analysis is then conducted in ANSYS-Fluent package by keeping time unsteady k-turbulence model. Air is used as the working medium the geometry (physical bounds) of the model is described. The quantity occupied via the fluid is divided into discrete cells (the mesh). The mesh may be uniform or non-uniform. Physical modeling is described for example; the equations of motions, enthalpy, radiation, species conservation boundary conditions are described. This involves specifying the fluid conduct and properties at the boundaries of the problem. For transient issues, the initial condition is also described. The simulation is run and the equations are solved iteratively as a regular or transient. Finally, a postprocessor is used for the analysis and visualization of the resulting solution. CFD can be used to decide the overall performance of a factor on the layout design, or it could be used to analyses difficulties with an existing component and result in its stepped forward design. For example, the strain drop via an element can be considered immoderate: the first step is to become aware of the region of interest: The geometry of the vicinity of interest is then defined. If the geometry already exists in CAD, it can be imported without directly. The mesh is then created. After importing the mesh into the preprocessor,

Table 1. Geometrical parameter of vortex tube.

Design Parameters	Dimensions (mm)	Material (steeliness steel)
Working tube length	520	-
Working tube inlet diameter (D)	50	-
Orifice diameter	50	-
Diameter of inlet nozzle Dn	50	-
Number of nozzles	4	-
Cold exit length	20	-
Hot exit area	500	-

different elements of the simulation including the boundary conditions (inlets, outlet, mass flow rate, etc.) and all the fluid properties like viscosity, density, thermal conductivity are described. The flow solver is administered to provide a record of results that comprise the version of speed, pressure, and other variables at some point of the place of interest. The results can be visualized and may provide the engineer and knowledge of the conduct of the fluid throughout the vicinity of behavior. This could cause modifications changes which can be examined by using converting the geometry of the CFD version and seeing the impact. The procedure of acting a single CFD simulation is break up into four additives. The path-lines representing the flow path are as shown in **Figure 3** the overall CFD results are drawn for the vortex tube with better performance in **Table 2**.

2.1. Experimental Set-Up

A test rig is designed and constructed in the laboratory where a compressed air outlet is located. Experiments were carried out at various operating conditions. To observe the experimental configuration of energy separation inside the vortex tube experimental tests have been designs by using the given parameters in **Table 1**. Ambient air is compressed by the compressor. the air coming from the compressor bypass through a pipe into the tube at first compressor was put on to get the compressed air at required pressure frequently from the receiver, the air is first injected into tube chamber passing through the nozzle the air input the tube with tangential velocity due to which create spiral inner in the tube. Digital Thermocouples used at the cold and hot exits of the vortex tube to measure the temperature. The thermocouples are coupled with the millimeters to take the temperature readings both temperatures at the hot and cold sides. Mass flow meters provided at the inlet and the hot outlet is used to measure the inlet mass flow rate and the hot outlet flow rate. A manometer is used to measure the stress and additionally measure. Volumetric drift meter is installed to calculate the mass flow with the flow fee of the compressed air. Airflow was allowed to continue the flow and temperature up to reach on steady-state. A series of experiments are carried out to analyze the performance of the system and to optimize the geometrical parameters. The experimental set up is given in **Figure 4**.

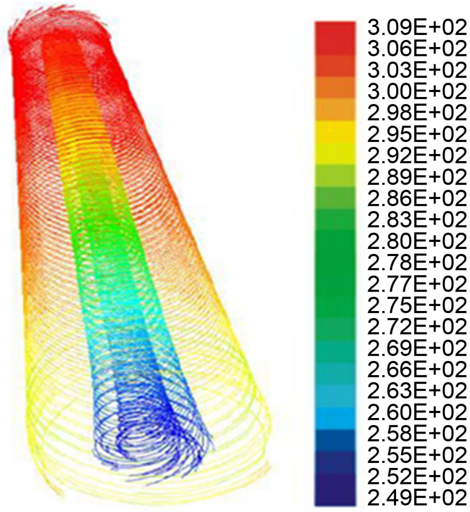

Figure 3. Total temperature along the vortex tube.

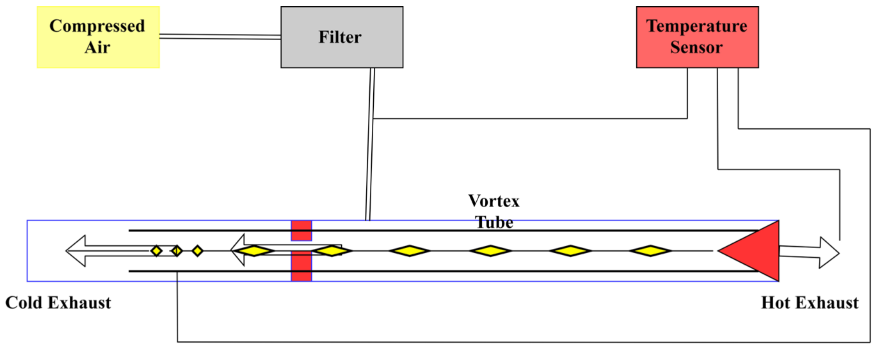

Figure 4. Experimental set up.

Table 2. CFD result.

CFD Results Number	Inlet pressure Pi (bar)	Cold Temperature Tc (˚C)	Hot Temperature Th (˚C)
1	6	16.99	30.50
2	8	14.30	32.02
3	10	13.19	34.05
4	12	11.60	36.10
5	14	9.00	39.88
6	16	7.55	41.20
7	18	5.70	43.54

Measuring Instruments Specification

In Table 3, the range of the digital thermocouple type is from −50˚C to 300˚C while accuracy is −20˚C to 120˚C is connected to both hot and cold side streams. Temperature readings of cold side and hot sides are observed by varying in the inlet pressure values from 18 bars. Manometer is used for pressure measuring range from 0 to 200 bars and volumetric flow meter is used for flow measuring. To determine the range of temperature difference for thermoelectric operation,

Table 3. Measuring instruments.

Sr. No.	Instrument	Range
1	Digital Thermocouple	−50˚C ± 300˚C
2	Manometer	0 - 200 bars
3	Volumetric flow meter	0 - 500 SLPM

the thermoelectric module was examined and the experimental result shown in Table 4.

3. Result and Discussion

Experimental analysis has been performed to examine the temperature difference. Various observations were received for specific parameters of the vortex tube, the hot and cold side temperatures for special inlet pressure. Table 4 which is given suggests the variant in ΔTc and ΔTh with the variation in any change of the following. Because the cone angle was reduced and better results had been acquired. As we develop the cone in the hot tube as much as sure level, we get the most efficient temperature variety. The angle became kept 11˚ for obtaining the most excellent effects is shown in Figure 5.

3.1. The Effect of Change in Inlet Pressure on Cool Temperature

The effect of change in cold temperature is along with the change in inlet pressure. It is observed that the temperature difference increased with the increase in the inlet pressure. For an inlet pressure of 18 bars, the cold temperature ΔTc changed as 3.70˚C. The figure shows that the impact of an Increase in ΔTc with an increase in inlet pressure is directly related to Pi is given in Figure 6.

3.2. The Effect of Change in Inlet Pressure on Hot Temperature

The impact of alternate hot temperature difference concerning with the change in inlet pressure. It seems to determine that the temperature difference improved with the mild difference with the change in inlet pressure. For an inlet pressure of 18 bar, the hot end temperature ΔTh was obtained as 42.09˚C. An increase in ΔTh with an increase in inlet pressure Pi is shown in Figure 7.

3.3. Effect on the Performance of Vortex Tube of the Nozzle

It is important to have a high peripheral velocity in the portion of the tube immediately after the nozzle. The nozzle curve affects the performance of vortex tube. The two conventional types of nozzles are a normal rectangle and Archimedes' spiral nozzles. In addition, the two types of conventional nozzle reach the peak of the cooling effect this improvement can make the nozzle gain a larger cooling rate per unit mass.

3.4. Effect of Hot Tube Length

The impact of hot tube length on temperature separation became observed. It

Table 4. Experimental results.

Experimental Results Sr. No.	Inlet pressure Pi (bar)	Cold Temperature Tc (˚C)	Hot Temperature Th (˚C)
1	6	15.93	33.05
2	8	13.52	34.84
3	10	11.26	35.69
4	12	9.27	37.10
5	14	6.70	39.30
6	16	4.66	41.22
7	18	3.70	42.09

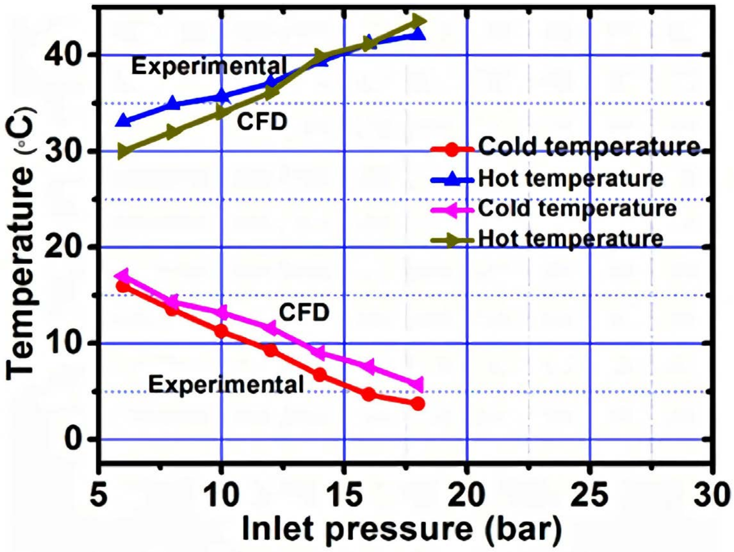

Figure 5. Comparison between experimental and CFD analysis.

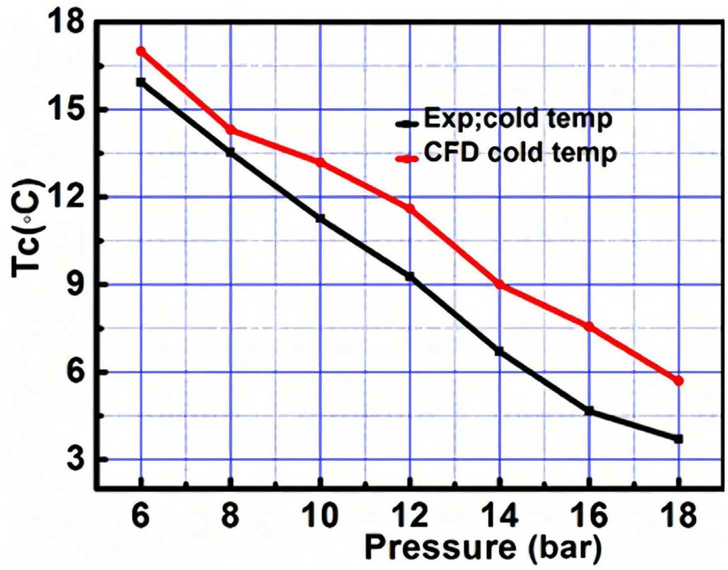

Figure 6. Increase in ΔTc with increase in inlet pressure Pi.

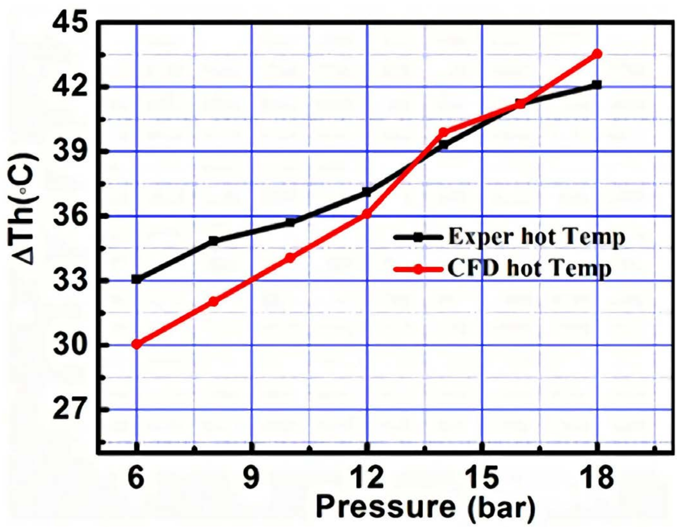

Figure 7. Increase in ΔTh with increase in inlet pressure Pi.

suggests the temperature gradients among the hot side and cold side outlet as a characteristic of cold mass frication, at specific aspect ratio for extraordinary lengths of vortex tube ratio, for variable tube period the inlet pressure and variety of nozzle changed into kept constant four nozzles respectively. It provided that the temperature distinction for various duration of warm tube has a relatively small difference. Experimental results indicate that the length of hot tube has little effect on temperature.

4. Conclusions

The energy separation in a vortex tube has been investigated by CFD and experimental routes. The important outcomes of this study are as follows:

1) Experimental results show that the highest temperature reduction will be obtained at the highest inlet pressure. It is found that the Inlet pressure is a key parameter for the vortex tube performance.

2) Increasing the inlet pressure improves the energy separation effect in general. However, when the ratio of inlet pressure to the cold outlet pressure ratio becomes large, then the rate of the cold outlet temperature decreases and levels are off. Furthermore, when this pressure ratio exceeds certain values so that shock waves appear, the temperature starts to decrease, corresponding to a transition from subsonic to supersonic flow at the tube inlet.

3) An optimization of the temperature drop with geometric parameters has been performed starting from the original design, and a large-scale optimized vortex tube providing the required performance is found. The temperature at the hot end increases with the decrease in the cone angle.

4) The vortex tube length to diameter ratio has a significant effect on the energy separation process. Nozzle has widely affected on vortex tube performance.

5) Comparing the experimental results with the obtained CFD results. The

CFD results were 8% higher than experimental additional confidence in the current model.

6) Experimental validation study was used to design a computational model of vortex tube with better thermal performance.

Acknowledgements

This work was supported by the China Postdoctoral Science Foundation (2017M623157), and Natural Science Foundation of Shaanxi Province (2018JQ5040).

Conflicts of Interest

The authors declare no conflicts of interest regarding the publication of this paper.

References

[1] Hamdan, M.O., Alsayyed, B. and Elnajjar, E. (2013) Nozzle Parameters Affecting Vortex Tube Energy Separation Performance. *Heat and Mass Transfer*, **49**, 533-541. https://doi.org/10.1007/s00231-012-1099-2

[2] Hilsch, R. (1947) The Use of the Expansion of Gases in a Centrifugal Field as Cooling Process. *Review of Scientific Instruments*, **18**, 108-113. https://doi.org/10.1063/1.1740893

[3] Hartnett, J.P. (1957) Experiment Study of the Velocity and Temperature Distribution in a High Velocity Vortex Type Flow. *Journal of Fluids Engineering*, **79**, 751-757.

[4] Ahlborn, B.K. and Gordon, J.M. (2000) The Vortex Tube as a Classic Thermodynamic Refrigeration Cycle. *Journal of Applied Physics*, **88**, 3645-3653. https://doi.org/10.1063/1.1289524

[5] Stephan, K., Lin, S., Durst, M., *et al.* (1983) An Investigation of Energy Separation in a Vortex Tube. *International Journal of Heat and Mass Transfer*, **26**, 341-348. https://doi.org/10.1016/0017-9310(83)90038-8

[6] Kurosaka, M. (1982) Acoustic Streaming in Swirling Flow and the Ranque-Hilsch (Vortex-Tube) Effect. *Journal of Fluid Mechanics*, **124**, 139-172. https://doi.org/10.1017/S0022112082002444

[7] Rattanongphisat, W. and Thungthong, K. (2014) Improvement Vortex Cooling Capacity by Reducing Hot Tube Surface Temperature: Experiment. *Energy Procedia*, **52**, 1-9. https://doi.org/10.1016/j.egypro.2014.07.048

[8] Behera, U., Paul, P.J., Dinesh, K., *et al.* (2008) Numerical Investigations on Flow Behavior and Energy Separation in Ranque-Hilsch Vortex Tube. *International Journal of Heat and Mass Transfer*, **51**, 6077-6089. https://doi.org/10.1016/j.ijheatmasstransfer.2008.03.029

[9] Gao, C.M., Bosschaart, K.J., Zeegers, J.C.H., *et al.* (2005) Experimental Study on a Simple Ranque-Hilsch Vortex Tube. *Cryogenics*, **45**, 173-183. https://doi.org/10.1016/j.cryogenics.2004.09.004

[10] Guen, M., Natkaniec, C., Kammeyer, J., *et al.* (2013) Effect of the Conical-Shape on the Performance of Vortex Tube. *Heat and Mass Transfer*, **49**, 521-531. https://doi.org/10.1007/s00231-012-1098-3

[11] Makode, M.V.A. (2014) Review on Experimental and CFD Analysis of Ranque-

Hilsch Vortex Tube. *International Journal of Engineering Research & Management Technology*, **2**, 333-337.

[12] Pourmahmoud, N., Zadeh, A.H., Moutaby, O., *et al.* (2012) Computational Fluid Dynamics Analysis of Helical Nozzles Effects on the Energy Separation in a Vortex Tube. *Thermal Science*, **16**, 151-166. https://doi.org/10.2298/TSCI110531085P

[13] Ahlborn, B. and Groves, S. (1997) Secondary Flow in a Vortex Tube. *Fluid Dynamics Research*, **21**, 73-86. https://doi.org/10.1016/S0169-5983(97)00003-8

[14] Khazei, H., Teymourtash, A.R. and Malek-Jafarian, M. (2012) Effects of Gas Properties and Geometrical Parameters on Performance of a Vortex Tube. *Scientia Iranica*, **19**, 454-462. https://doi.org/10.1016/j.scient.2012.03.003

[15] Joseph, R.G. (1934) Method and Apparatus for Obtaining from Alpha Fluid under Pressure Two Currents of Fluids at Different Temperatures. U.S. Patent 1,952,281.

[16] Sahu, R., Bhadoria, R. and Patel, D. (2012) Performance Analysis of a Vortex Tube by Using Compressed Air. *International Journal of Scientific and Engineering Research*, **3**, 1-7.

[17] Secchiaroli, A., Ricci, R., Montelpare, S., *et al.* (2009) Numerical Simulation of Turbulent Flow in a Ranque-Hilsch Vortex Tube. *International Journal of Heat and Mass Transfer*, **52**, 5496-5511. https://doi.org/10.1016/j.ijheatmasstransfer.2009.05.031

[18] Rattanongphisat, W., Riffat, S.B. and Gan, G. (2008) Thermal Separation Flow Characteristic in a Vortex Tube: CFD Model. *International Journal of Low Carbon Technologies*, **3**, 282-295. https://doi.org/10.1093/ijlct/3.4.282

[19] May, O., Mokni, I., Mhiri, H., *et al.* (2011) CFD Investigation of a Vortex Tube: Effect of the Cold End Orifice in the Temperature Separation Mechanism. *Science Academy Transactions on Renewable Energy Systems Engineering and Technology*, **1**, 84-89.

[20] Bramo, A. and Pourmahmoud, N. (2010) A Numerical Study on the Effect of Length to Diameter Ratio and Stagnation Point on the Performance of Counter Flow Vortex Tube. *Australian Journal of Basic and Applied Sciences*, **4**, 4943-4957.

[21] Fröhlingsdorf, W. and Unger, H. (1999) Numerical Investigations of the Compressible Flow and the Energy Separation in the Ranque-Hilsch Vortex Tube. *International Journal of Heat and Mass Transfer*, **42**, 415-422. https://doi.org/10.1016/S0017-9310(98)00191-4

[22] Farouk, T. and Farouk, B. (2007) Large Eddy Simulations of the Flow Field and Temperature Separation in the Ranque-Hilsch Vortex Tube. *International Journal of Heat and Mass Transfer*, **50**, 4724-4735. https://doi.org/10.1016/j.ijheatmasstransfer.2007.03.048

[23] Skye, H.M., Nellis, G.F. and Klein, S.A. (2006) Comparison of CFD Analysis to Empirical Data in a Commercial Vortex Tube. *International Journal of Refrigeration*, **29**, 71-80. https://doi.org/10.1016/j.ijrefrig.2005.05.004

[24] Rao, K.K.K., Ramesh, A., Rajesh, M., *et al.* (2016) Experimental Analysis of Vortex Tube by Varying the Geometry and Material—A Review. *Journal of Mechanical Engineering and Biomechanics*, **1**, 1-13.

[25] Dutta, T., Sinha Mahapatra, K.P. and Bandyopadhyay, S.S. (2010) Comparison of Different Turbulence Models in Predicting the Temperature Separation in a Ranque-Hilsch Vortex Tube. *International Journal of Refrigeration*, **33**, 783-792. https://doi.org/10.1016/j.ijrefrig.2009.12.014

[26] Singh, G. and Sharma, V.S. (2017) Analyzing Machining Parameters for Commercially Pure Titanium (Grade 2), Cooled Using Minimum Quantity Lubrication As-

sisted by a Ranque-Hilsch Vortex Tube. *International Journal of Advanced Manufacturing Technology*, **88**, 2921-2928. https://doi.org/10.1007/s00170-016-8982-9

[27] Prabhakaran, J., Vaidyanathan, S. and Ka Nagarajan, D. (2012) Establishing Empirical Relation to Predict Temperature Difference of Vortex Tube Using Response Surface Methodology. *Journal of Engineering Science and Technology*, **7**, 722-731.

[28] Kumar, G.S., Padmanabhan, G. and Sarma, B.D. (2016) Simultaneous Optimization of Hot and Cold Outlet Air Temperatures of Vortex Tube Using Grey Relational Analysis and Taguchi Method. *Journal of Engineering Science and Technology*, **11**, 1543-1553.

[29] Pinar, A.M., Uluer, O. and Kırmacı, V. (2009) Statistical Assessment of Counter-Flow Vortex Tube Performance for Different Nozzle Numbers, Cold Mass Fractions, and Inlet Pressures via Taguchi Method. *Experimental Heat Transfer*, **22**, 271-282. https://doi.org/10.1080/08916150903099058

[30] Kshirsagar, O.M., Ankolekar, V.V. and Kapatkar, V.N. (2014) Effect of Geometric Modifications on the Performance of Vortex Tube—A Review. *International Journal of Engineering Research and Applications*, **1**, 92-98.