SIMULATION OF VORTEX TUBE USING NATURAL GAS AS WORKING FLUID WITH APPLICATION IN CITY GAS STATIONS

Jafargholinejad S.* and Heydari N. **
*Author for correspondence
*Department of Mechanical Engineering, Ardabil Branch, Islamic Azad University, Ardabil, Iran,
E-mail: s.jafargholinejad@iauardabil.ac.ir
**Department of Mechanical Engineering, Jolfa International Branch, Islamic Azad University, Jolfa, Iran,

ABSTRACT

The work has been done on a 3D model of vortex tube and the governing equations have been solved using ANSYS–FLUENT^{TM} software. Simulations were performed using density based solver and k-ε turbulence model. After validating some results of present study with available data, the effects of geometrical parameters and air/natural gas flow in a vortex tube with 6 and 2 tangential nozzles was examined. The performance of vortex tube with 6 nozzles was better when compared with 2 nozzles which is in satisfactory agreement with previous experiments, so we only show the results for the 6 nozzles vortex tube. There is a little difference between air and natural gas as a working fluid. We found that the temperature difference between hot and cold ends for air flow was little more than natural gas. Based on the results obtained in this work it is concluded that vortex process based on pressurized natural gas can be used in C.G.S as a heat exchanger for high efficiency operation and energy saving purposes.

INTRODUCTION

Vortex tube is a device, which can separate a higher pressure input gas into two lower-pressure flows with different temperatures. One flow has a higher temperature than the input gas temperature while the other has a temperature lower than that. The vortex tube has numerous advantages such as its low volume, ease of production and maintenance, a high safety factor, lack of any movable parts, and simple structure. Among the experimental studies, results of the work by Cockrill [1] indicated that the performance and efficiency of counter-flow vortex tubes are better than those of the uni-flow vortex tubes. Figure (1) depicts a counter-flow (or standard) vortex tube.

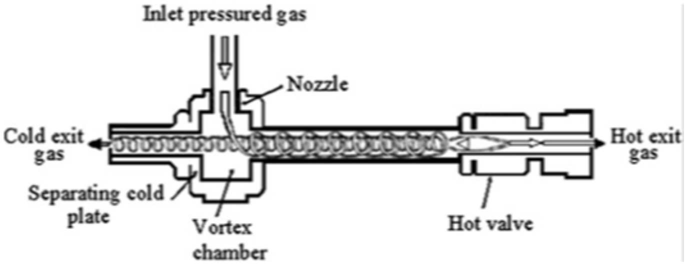

Figure 1 standard counter-flow vortex tube

In this research, a counter-flow vortex tube (or a standard vortex tube) was used. This device was discovered by Ranque, the French physicist and metallurgist [2]. Several years later, studies of vortex tubes were continued by a German scientist named Hilsch [3]. After Hilsch, broad experimental and theoretical research was conducted on vortex tubes by various scientists and researchers. Ahlborn et al. [4] examined flow inside a vortex tube. They considered the presence of a secondary swirl flow in a vortex tube. **C**omparison between the performance predicted by a computational fluid dynamic (CFD) model and experimental measurements taken using a commercially available vortex tube reported by Skye [5]. Other researchers such as Selek [6] and Dincer [7] conducted experimental research on flow and temperature in a vortex tube. Aljuwayhel et al. [8] used a numerical model to examine the principles of energy separation in a vortex tube. Frohlingsdorf [9] modelled flow in a vortex tube using a CFX code. Khait et al. [10] studied air flow in a vortex tube. They used various models to simulate turbulent flow. Oliver et al. [11] modeled turbulent air flow inside a Ranque–Hilsch vortex tube in three dimensions using ANSYS-CFX. Their findings stress the significance of meshing especially in the cold region. Zein et al. [12] studied the effect of the ratio of vortex tube length to diameter on simulation of vortex tube flow. According to their findings, the inertia point inside a vortex tube is independent of vortex tube length. Reynolds [13], and Deissler and Perlmutter [14] considered energy separation a result of friction and turbulence.

There are always limitations on experimental studies which result from limitations of test devices. However, with Computational Fluid Dynamics (CFD) it is possible to eliminate these limitations. One of the CFD studies on pressure is the study by Ameri and Behnia [15], who indicated that with an increase in the inlet pressure the efficiency decreases. Poshernev and Khodorkov [16] used natural gas as the working fluid and turned it into liquid using a vortex tube. Comparison between the performance predicted by a computational fluid dynamic (CFD) model and experimental measurements taken using a commercially available vortex tube

In this research, the fluid flow is analysed inside a vortex tube and the effects of various factors (such as geometry, inlet gas boundary conditions, and size of inlet gas nozzle) on performance of vortex tube is studied to determine the appropriate conditions for performance of the vortex tube. Distribution of temperature is analysed as the main variable, and distribution of velocity is studied as another variable. This is because temperature distribution in a vortex tube depends on velocity distribution. Therefore, the system dimensions and dimensionless parameters, which are dependent on system dimensions, are selected such that feasibility of application of this system as a heater in the C.G.S (City Gate Station) is studied in practice. For this purpose, numerical inputs of the software, in which the three-dimensional geometry of the vortex tube is implemented, are obtained from the real data of the C.G.S station (e.g. supply air/gas pressure). In addition to thermophysical characteristics of air, the specifications of a gas like methane are put into the simulation software and results are compared.

Natural gas pressure regulating stations at the inlets of the C.G.S. urban gas network reduce the gas pressure in the transmission line from about 1050 psi to about 250 psi. The process of reducing pressure occurs in the regulator and leads to a drastic decrease in gas temperature. A decrease in gas temperature may result in condensation and even freezing of water vapour in the gas. Consequently, there is a possibility of blockage of the gas duct. Therefore, using indirect water bath heaters the gas temperature is increased partially before it enters the regulator to prevent freezing. The method commonly used in pressure regulating stations is by heating the gas flowing into the regulator to a temperature of about 40. Today, in all pressure regulating stations the natural gas heaters are employed. These heating systems burn some natural gas to heat natural gas and warm it before it enters the regulator and to prevent freezing at the time of pressure decline.

The vortex tube can be installed in natural gas pressure regulating stations to heat the gas in the regulator pilots. The objective is to optimize consumption of energy by using vortex tube in the C.G.S station and reduce gas heating costs. The importance of this technique is considerably significant. For example, in a C.G.S station with a fumigation capacity of 50000 m3/h, the average amount of gas used by a two-flame heater to heat and prevent freezing of a regulator is about 400 m3/h per a day. With the use of vortex tubes and energy of the pressure regulating system energy will be saved considerably. In addition we can use the cold end flow in a refrigeration cycle for air conditioning purposes or converting compressed natural gas (CNG) to liquefied natural gas (LNG).

NOMENCLATURE

a	[mm]	Cold exit diameter
c	[mm]	Cold exit length
CF	[-]	Cold fraction
C_p		Constant pressure heat capacity
d	[mm]	Nozzle width
D	[mm]	Hot exit diameter
E	[mm]	Coniacal valve length
F	[mm]	Conic base diameter
G	[mm]	Gap lebgth
k	[m^2/s^2]	Turbulence kinetic energy
L	[mm]	Working tube length
P	[N/m^2]	pressure
t	[s]	time
T	[K]	Temperature
u	[m/s]	Velocity
Special characters		
ε	[m^2/s^3]	Turbulence dissipation rate
μ	[kg /m.s]	Dynamic viscosity
μ_t	[kg /m.s]	Eddy viscosity

GEOMETRY DESCRIPTION

The computational domain chosen for current investigation is shown in Fig. 2. The geometric parameters and values are shown in Fig. 3 and table 1 respectively.

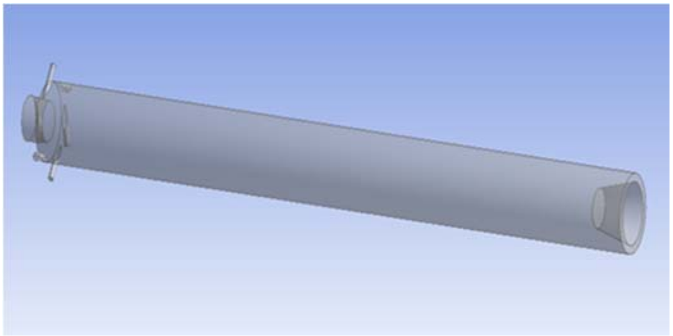

Figure 2 Geometrical shape of the 6 nozzle vortex tube

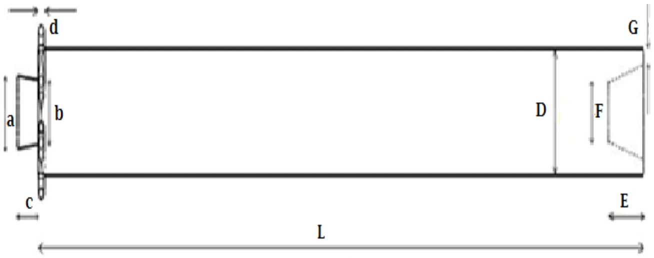

Figure 3 Geometric parameters of the vortex tube

Table 1 Geometric values of the vortex tube

parameter	Value(mm)
a	**6.2**
b	**5.08**
c	**4**
d	**1.1**
D	**11**
E	**6.5**
F	**5.24**
G	**0.84**
L	**106**

GOVERNING EQUATIONS

The compressible turbulent and high rotating flow inside the vortex tube is assumed as three-dimensional, steady state and the standard $K-\varepsilon$ turbulence model is employed.

The governing equations are arranged by the conservation of mass, momentum and energy equations, which are given by:

$$\frac{\partial \rho}{\partial t} + \nabla.(\rho u) = 0 \qquad (1)$$

$$\frac{\partial(\rho u)}{\partial t} + \nabla.(\rho u u) = -\nabla \rho + \mu \nabla^2 u \qquad (2)$$

$$\rho C_P[\partial T/\partial t + \nabla.(uT)] = K\nabla^2 T \qquad (3)$$

The effect of turbulence present in the flow is captured by using standard k-ε turbulence model where the eddy viscosity of momentum equation is modeled by:

$$\mu_{t=C_\mu}\, \rho \left(\frac{k^2}{\varepsilon}\right) \qquad (4)$$

Where k, ε and C_μ represents turbulence kinetic energy, turbulence dissipation rate and a constant equals to 0.09, respectively.

Here we define an important parameter, Cold fraction (CF) which will be used in the analyzing the simulation results. CF can be defined as the amount of the gas exiting through the cold end of the vortex tube to the pressurized gas input. The cold mass fraction can be controlled by the cone valve, which is placed at the hot end of the tube.

NUMERICAL METHOD

Present work has been executed through finite volume based solver of ANSYS FLUENT^{TM} 16.0 for solving the governing equation and implementing the boundary condition. A solid model of the vortex tube with six inlet nozzles has been created in ANSYS design model software.

To generate the grid for the desired geometry ANSYS Meshing software was employed. Fig. 4 shows the hexagonal elements used for the three-dimensional model of the vortex tube. To obtain more precise solutions, the sensitive and fine geometrical areas are meshed with smaller dimensions.

Simple algorithm was used to couple velocity and pressure. Due to the compressibility of flow the density-based solver was used in this problem. The density-based solver solves the mass, momentum, energy, and transmission equations through coupling.

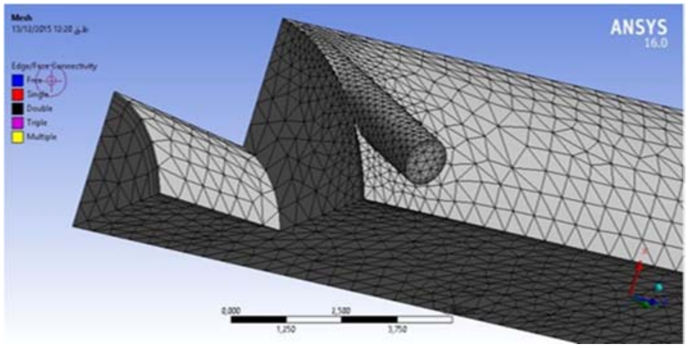

Figure 4 Refinement of meshes near the critical regions of vortex tube in sector

Validation of Numerical Models

In this section, to ensure the accuracy of the simulation and validation phases, the results are examined with a previous powerful work [5].

For instance, the approximate results of our work and comparison of results are shown at several points in table 2.

The Skye et al. [5] CFD model was developed in a two dimensional form, but the present one is three dimensional and both models use similar geometry and boundary conditions. It is shown in this table that our results is in good agreement with the corresponding computational and experimental values of previous work.

Table 2 Comparison of results with previous work

Cold Fraction	Skye et al.[5] Experimental Results[K]	Skye et al.[5] Computational Results[K]	Present Work Results[K]
0.3	16	12	17
0.5	31	25	28
0.7	55	45	38

(a) Hot Temperature Separation

Cold Fraction	Skye et al.[5] Experimental Results[K]	Skye et al.[5] Computational Results[K]	Present Work Results[K]
0.3	42	28	28
0.5	37	26	28
0.7	29	21	19

(b) Cold Temperature Separation

Fig. 5 and Fig. 6 show the temperature separation as a function of the cold fraction for airflow in vortex tube. The temperature separation is the absolute difference between the inlet temperature and the respective exit temperatures. As can be seen in these figures the data obtained in the present work are in good agreement with previous work by Skye et al. [5].

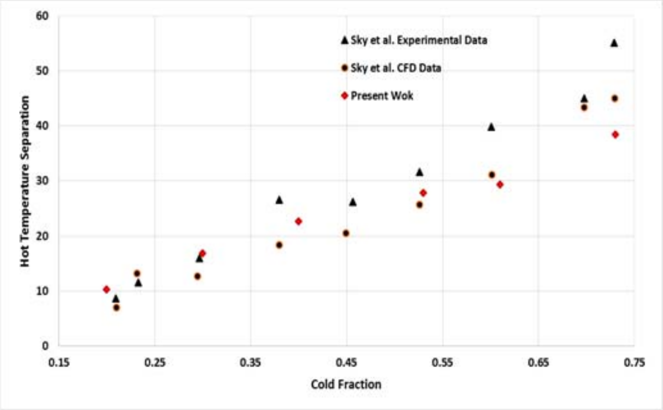

Figure 5 Hot temperature separation as a function of cold fraction.

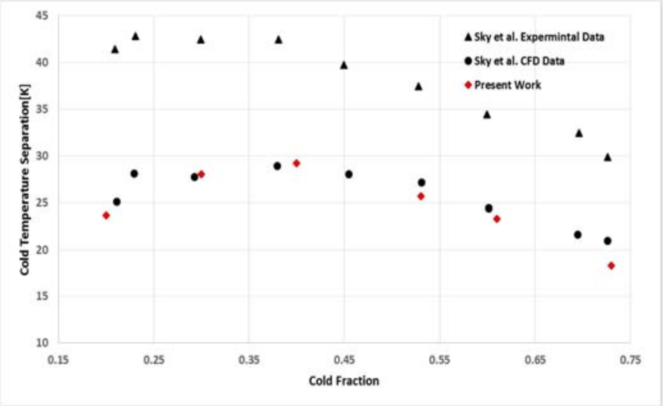

Figure 6 Cold temperature separation as a function of cold fraction.

RESULTS

First of all the grid independence study has been accomplished for CF=0.5. Present work uses unstructured mesh with boundary layer mesh. Difference between the cold and hot output temperatures has been used as a parameter to check grid sensitivity of the computed results. The numbers of cells are varied from coarse to fine in the range of 20000 to 120000. Figure 7 shows variation of difference between the cold and hot ends temperatures of vortex tube with number of cells used in computational domain. It can be seen that with increasing fineness of computation domain, cold temperature reach to an optimum number. So all computations thereafter were executed using 100000 cell computation domain.

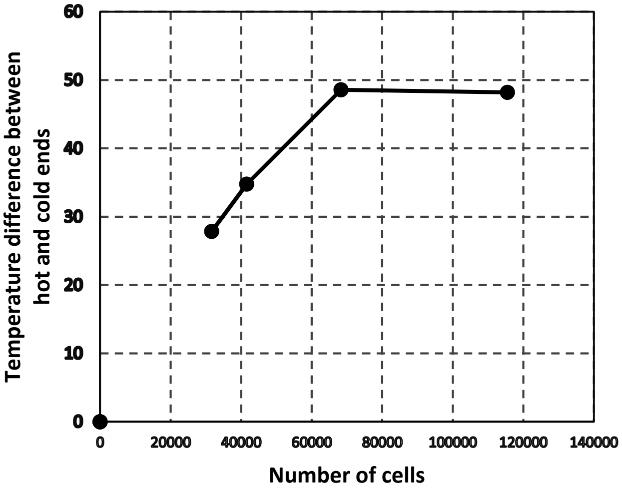

Figure 7 Grid size independence study

The flow and thermal structure emerging inside the vortex tube is illustrated through 3-D streamlines with temperature color code is shown in Fig. 8. Development of peripheral and core vortex moving can be observed from this figure.

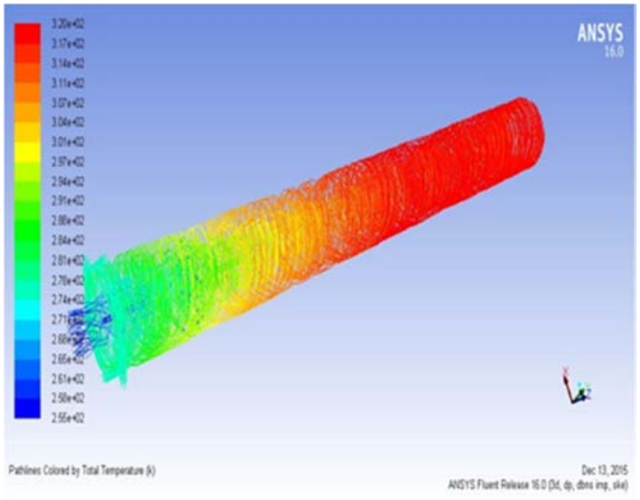

Figure 8 Streamlines coloured with temperature variation

Figure 9 shows the results obtained for methane gas flow in a vortex tube. Simulation are performed under one of the Ardabil city gas station conditions. For this purpose, the methane gas flows into the vortex tube at temperature of 277.6 K and pressure of 17 bar and 45 bar respectively.

Results suggest that the range of variations in cold output temperature at input pressures of 17 and 45 bar are similar, but the range of warm end temperature is higher at an input pressure of 45 bar. The maximum temperature 320.2 K observed for the warm output was obtained at the pressure of 45 bar. The minimum cold output temperature of 247.2 K was obtained at the pressure of 17 bar.

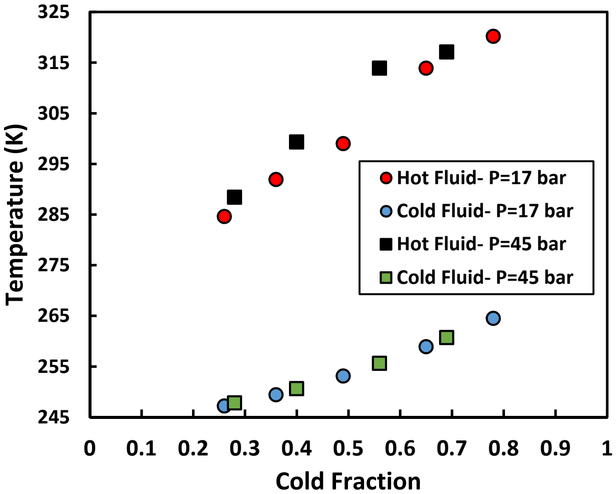

Figure 9 Total temperature as a function of cold fraction for methane gas under the C.G.S condition

Variations in temperature along the tube for CF = 0.49 are depicted as contours in the figure 10. It can be seen that peripheral flow is warmer than the core flow. This phenomena is called energy separation in vortex tube because there are no heat or work inputs to the device.

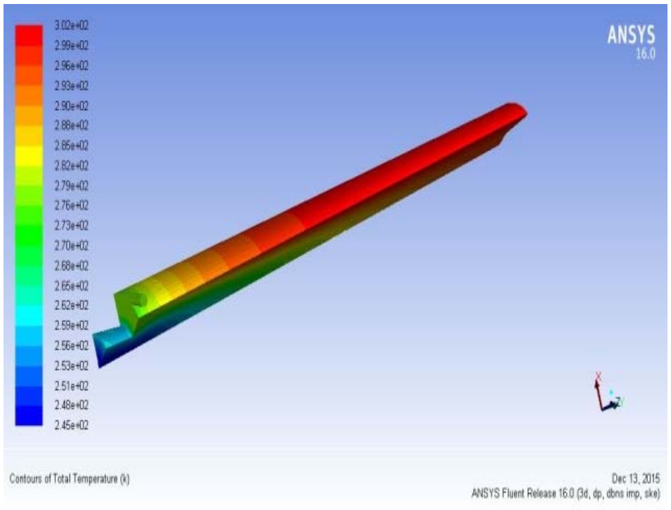

Figure 10 Total temperature distribution in a vortex tube for methane gas in CF=0.49

Difference between the temperature distribution of air and methane flow through a vortex tube is shown in Fig. 11. As can be seen in this figure there in a little difference between two fluids. For the range of cold fraction hot end temperature for air flow is slightly more than methane flow and cold end temperature for air flow is slightly lower than the methane gas flow.

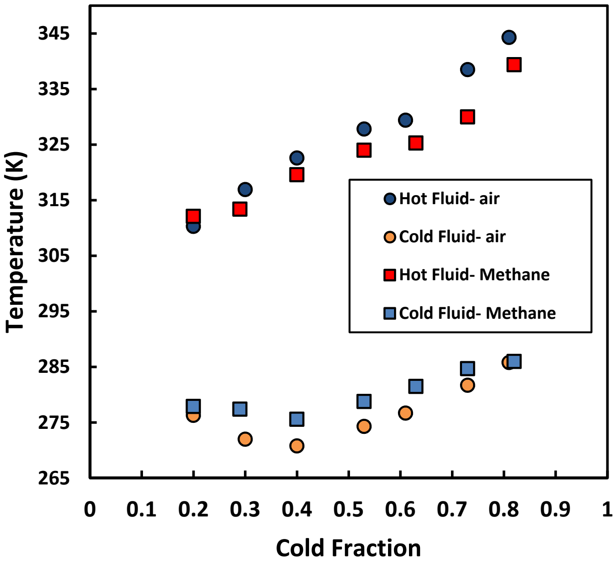

Figure 11 Comparison of Total temperature as a function of cold fraction for air and methane gas flow through a vortex tube

CONCLUSION

In this research, the results were obtained based on numerical modelling of methane gas flow through a 6 nozzle vortex tube under city gas station conditions. To obtain the optimum and effective performance of a vortex tube, the suitable length to diameter and tube length to nozzle diameter ratios are selected.

Based on the obtained results it is concluded that for a wide range of cold fraction, an increase in the cold fraction results in an increase in the warm output gas temperature. In addition almost for all of the values of the cold fraction, temperature difference between the input fluid and exiting fluid are higher for air flow than methane gas flow. In fact, the air flow results in a slightly higher warm output temperature and lower cold output temperature.

For the methane gas flow The ranges of variations of cold output temperature at 17 and 45 bar input pressures are almost similar, but the warm output temperature range is slightly higher at an input pressure of 45 bar.

Vortex tube is an energy saving device, which could take the place of heaters in city gas stations for anti-freezing purposes. Simultaneously it can be used in the refrigeration cycle for liquefaction of compressed natural gas and converting it to liquefied natural gas.

REFERENCES

[1] Cockerill T., Ranque-Hilsch vortex tube. M.Sc Thesis, University of Cambridge., 1995.

[2] Ranque.G.J, Experiences sur la détente giratoire avec productions Simultanes d'un echappement d'air chaud et d'un echappement d'air froid,J.phys.Radium, 4, 112-114, 1933.

[3] Hirsch R., The Use of the Expansion of Gases in a Centrifugal Field as a Cooling Process, Review of Scientific Instruments, 18(2) , 1947, pp.108–113

[4] Ahlborn B., Groves S., Secondary Flow in Vortex Tube, progress in Fluid Dynamics Research, 21, 73-86, 1997.

[5] Skye H.M., Nellis G.F. and Klein,S.A., Comparison of CFD Analysis to Empirical Data in a Commercial Vortex Tube, Int. J. Refrig., 29, 2006, pp. 71-80.

[6] Selek M., Tasdemir S., Dincer K. and Baskaya S., Experimental Examination of the cooling performance of Ranque-Hilsch vortex tube on the cutting tool nose point of the turret lathe through infrared thermography method, International journal of refrigeration, 1-9, 2010.

[7] Dincer K., Avci A., Baskaya S. and Berber A., Experimental investigation and energy analysis of the performance of a counter flow Ranque- Hilsch vortex tube with regard to nozzle cross-section areas, International journal of refrigeration, No. 33, 954-962, 2010.

[8] Aljuwayhel N.F., Nellis. G.F. and Klein S.A., Parametric and internal study of the vortex tube using a CFD model, International Journal of Refrigeration, 28,442-450, 2005.

[9] Frohlingsdorf W., H. Unger H., Numerical investigation of the compressible flow and the energy separation in the Ranque-Hilsch vortex tube, International Journal of Heat and Mass Transfer, 42,415-422, 1999.

[10] Anatoliy Khait, Alexander Noskov, Vladimir Alekhin & Aleksey Antipin, Numerical Simulation and Visualization of air Flow in Ranque-Hilsch Vortex Tube,, Proceedings of the 13th International Conference on Construction Applications of Virtual Reality, 30-31 October 2013.

[11] Oliver R, Boyle F, Reynolds A, Computer Aided Study of the Ranque–Hilsch Vortex Tube Using Advanced Three-Dimensional Computational Fluid Dynamics Software, Proceedings of the 6th WSEAS International Conference on Applied Computer Science, Tenerife, Canary Islands, Spain, December 16-18, 2006.

[12] Zin K.K, Hansske A. and Ziegler F., Modeling and Optimization of the Vortex Tube with Computational Fluid Dynamic Analysis, Energy Research Journal 1 (2): 193-196, 2010.

[13] Reynolds, A.J., A Note on Vortex-tube Flow, J.Fluid Mech., 14, 1962, pp.18-20.

[14] Deissler R.G., and Perlmutter M., Analysis of the flow and energy separation in a turbulent vortex, Int. J. Heat Mass Transfer, 1, 1960, pp. 173–191.

[15] Ameri, M. and Behnia B., The study of Key Design Parameters Effects on the Vortex Tube Performance, Journal of Thermal Science, 4, 2009, pp.370-376.

[16] Poshernev N.V., and Khodorkov I.L., Natural-gas tests on a conical vortex tube (CVT) with external cooling, Chemical and Petroleum Engineering, 40(3-4), 2004, pp. 212–217.