[1.] *Penka ZLATEVA*, [2.] *Krystin YORDANOV*, [3.] *Diyan M. DIMITROV*

# NUMERICAL SIMULATION OF TEMPERATURE SEPARATION PHENOMENA INSIDE VORTEX TUBE

[1,2.] *Technical University of Varna, Dep. Heat Engineering, BULGARIA*
[3.] *Technical University of Varna, Dep. Technical Mechanics, BULGARIA*

**Abstract:** *The objective of present article is an atempt to simulate numerically temperature separation phenomena (Ranque-Hilsch affect) inside the vortex tube. For this simulation computational fluide dynamic (CFD) analysis, applying standard k-ε turbulence model, is used. Geometry is drawn in 3D space using Autodesk Inventor Professional and transferred in Autodesk Simulation CFD to solve the problem. Initial conditions for inlet air stream: temperature 22ºC, flow rate 40l/min and gauge pressure 6 bar are set. As a result air flow velosity field and temperature field is found. Predicted temperatures at cold and hot end are 4 and 42ºC. It can be concluded that vortex tube can be analysed and further optimisated by CFD modeling.*

**Keywords:** *Ranque-hilsch vortex tube, temperature separation, CFD model, k-ε model, numerical simulation*

## 1. INTRODUCTION

The Vortex Tube (VT) is a device that generates cold and hot air from compressed air.It was invented by Georges J. Ranque in 1933[1]. The German engineer Rudolf Hilsch suggested the working vortex tube design in 1947 [2]. It is a simple low cost device, with no moving parts, no electricity and there is a possibility to adjust the temperature [3,4]. The main drawbacks are connected with its low thermal efficiency, the noise and the availability of compressed air. VT is commonly used for inexpensive spot cooling, when compressed air is available, like cutting tools of CNC machines, laboratory equipment, etc.

The vortex tube phenomena can be observed physically, but is dificult to explain. Compressed air is sent through the tangential inlet nozzle to create the vortex motion inside the tube, fig. 1.
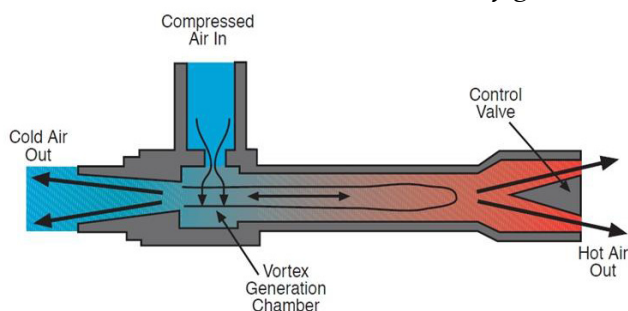


*Figure 1. Vortex tubeschematic [3]*

As the vortex moves along the tube, a temperature separation is formed. Hot air moves along the tube periphery, and cold air is in motion in the inner core. This resulting radial temperature separation inside the vortex tube is also called the Ranque-Hilsch effect. So far, there is no single theory that explains the radial temperature separation.

Nowadays, researches use computational fluide dynamic (CFD) to explain this phenomena [5-9]. In [5] the fluid dynamics model of the vortex tube is set in order to understand the temperature separation phenomena. In [6] the results from similar to [5] CFD model with commercially available vortex tube geometry is set. Researsches [5] also measure the temperatures at the hot and cold end of the VT and compare them with model predictions. The temperatures are in reasonable agreement with the experimental ones, if a standard k-ε turbolence model is used. In [7] a similar conclusion is given.

The aim of the present study is to createnumerical model of vortex tube effect in order to use it in a further optimisation of the design.

## 2. THE MODELING APPROACH

A computational fluid dynamic (CFD) analysis, using k-ε turbolence model, is set to solve fluid velocity and temperature field inside a vortex tube.

For this purpose, a commercially available software Autodesk Inventor Professional and Autodesk Simulation CFD, are used. The following approach is used to create the present CFD model.

## 2.1. Geometry conditions

The geometry of the vortex tube is designed in a way, which allows its parts to be made with universal metal cutting machines.It is modeled in 3D space as a solid body with the shape and dimensions shown on fig. 2. The swirl chamber is designed with two inlet nozzles. The inner diameterof 8 mm and L/D =30 were chosen. Such L/D ratio is suggsted in [8] for VT with similar design. Dimensions of the tube body are 385 x 30 x 54 mm.

To create CFD model this 3D geometry is transferred to Autodesk Simulations CFD environment.
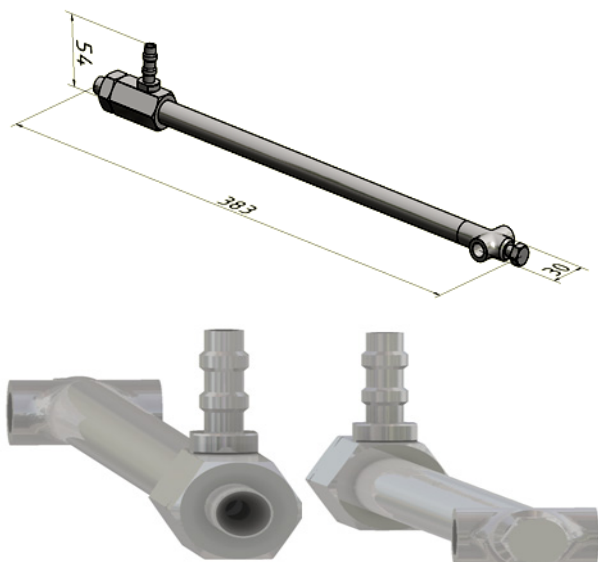




*Figure 2. 3D CAD model of vortex tube*

## 2.2. Physical conditions

The body of VT will be made of steel. To keep the price low, steel C45 is chosen as a VT material. The working fluid is air. All the cavities in VT are set as an air domain. The thermal properties of the used materials as a function of the temperature $c=c(t)$, $k=k(t)$ are taken from the sofware material library. In table 1 only values for temperature 22°C are given.

*Table 1.Properties of materials at 22°C*

| Materials | $C$, kJ/(kg.K) | $k$, W/(m.K) | $\rho$, kg/m³ |
|-----------|----------------|--------------|---------------|
| Air | 1,0057 | 258 | 1,205 |
| Steel C45 | 46,5 | 544 | 7,833 |

## 2.3. Initial and boundary conditions

The initial and boundary conditions, given at different areas to simulate the vortex tube, are as follows.

Pressure inlet boundary condition to the inlet area of the vortex tube (at the inlet of the nozzle) with volume flow rate of 40 l/min, total gauge pressure of 6 bar and total temperature of 22°C.

Pressure outlet boundary condition to the cold end region of vortex tube with gauge pressure of 0 bar.

Pressure outlet boundary condition to the hot end area of vortex tube with gauge pressure of 1 bar. Pressure at hot outlet is kept higher than that at the cold exit to reduce the flow through hot exit.

## 2.4. Meshing

The created mesh consists tetragonal finite elements. The size of the elements is 0.5 mm, figure 3. Their number is 103651, and the nodes are 27620.
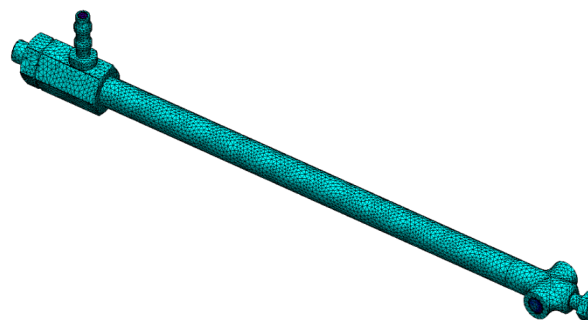


*Figure 3. Mesh of finite elements*

## 2.5. Problem solving

The created numerical model is solved in Autodesk Simulation CFD enviroment. The duration of the simulation is 220s, time steps 10s.

## 3. RESULTS AND ANALYSIS

As a result of the numerical simulation, the velocity and temperature field inside the vortex tube, are visualised. The following figures illustrate the fluid flow and temperature field at 55, 110, 165 u 220s, afterwards the fluid flow and temperature difference betweeen hot and cold end remain practically constant.

Figures 4-7 illustrate the distribution of the velocity magnitude in YZ plane. Figures 8 and 9 a,b illustrate the air velocity magnitude in XZ plane.

As it can be seen (fig. 4-7) the air in the core of the tube cross section has higher velocity and the air velociy increases at the cold end of the VT. Figures 8,9 show the vortex creation.
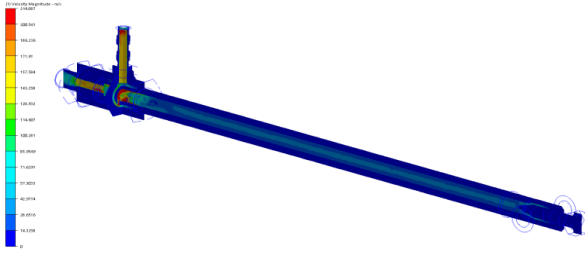
*Figure 4. Distribution of velocity magnitude
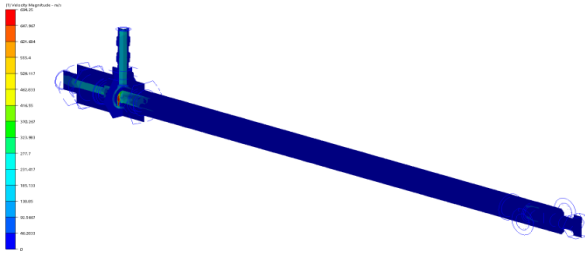at 55 s in the YZ plane [m/s]*



*Figure 5. Distribution of velocity magnitude
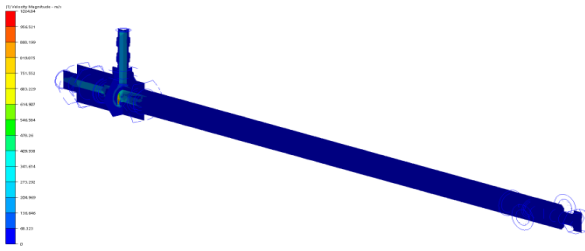at 110 s in the YZ plane [m/s]*



*Figure 6. Distribution of velocity magnitude
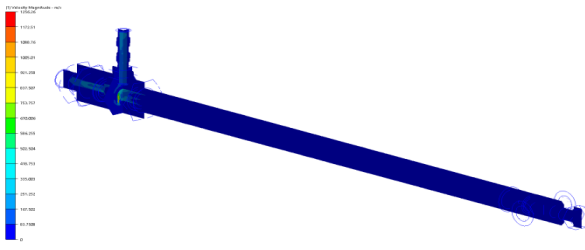at 165 s in the YZ plane [m/s]*



*Figure 7. Distribution of velocity magnitude
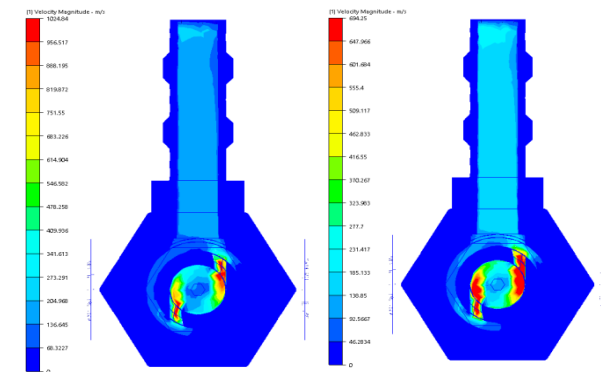at 220 s in the YZ plane [m/s]*



*a) at 55s*                    *b) at 110s*
*Figure 8. Distribution of velocity magnitude
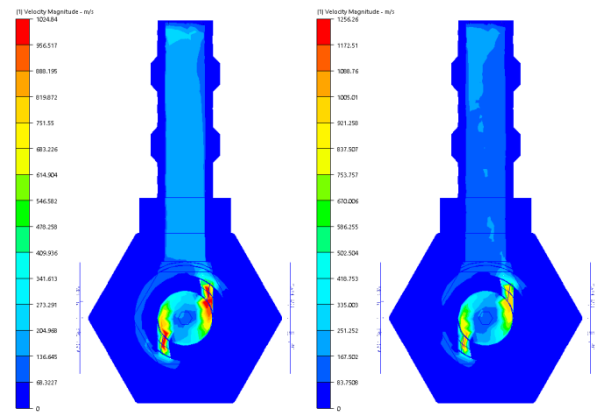in the XZ plane [m/s]*



*a) at 165s*                    *b) at 220s*
*Figure 9. Distribution of velocity magnitude
in the XZ plane [m/s]*

The temperature field, at the above mentioned time
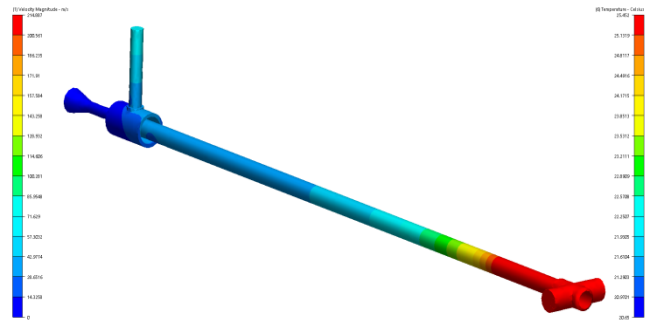intervals, is illustrated on figures 10-13.



*Figure 10. Distribution of temperature
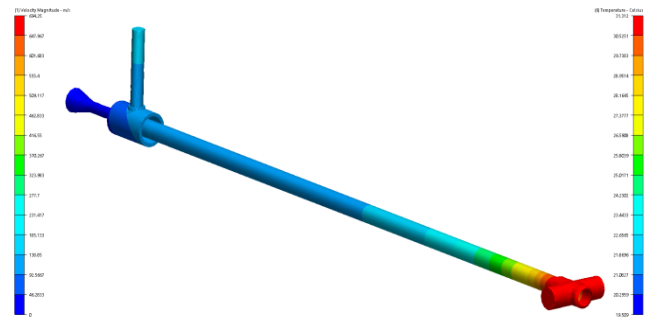at 55s (only air domain)*



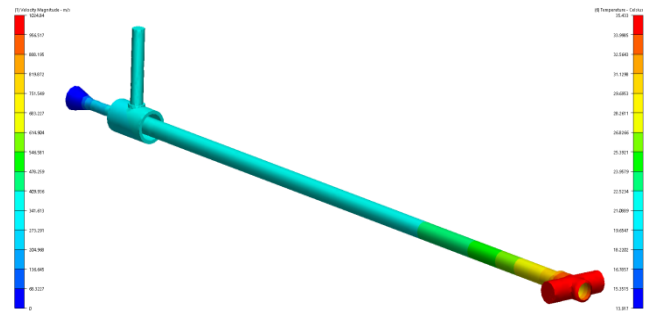*Figure 11. Distribution of temperature
at 110s (only air domain)*



*Figure 12. Distribution of temperature
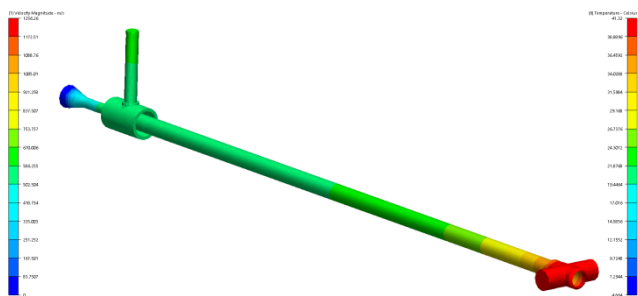at 165s (only air domain)*

*Figure 13. Distribution of temperature
at 220s (only air domain)*

The analysis of the above presented figures shows that if at the inlet of the designed vortex tube there is air with temperature of 22°C, a flow rate of 40l/min and total gauge pressure of 6 bar, a temperature change at the two ends of the vortex tube will be observed. The longer end is with a higher temperature than the one set at the inlet (hot end), and the shorter one is with a lower temperature than the one set at the inlet (cold end). Predicted temperatures at the ends after 220s are 4°C at the cold end and 42°C at cold one.

## 4. CONCLUSIONS

In conclusion, it is shown that the CFD approach is applicable for simulating Ranque-Hilsch effect. The details of the flow velocity and temperature distribution are presented. It is found that various levels of complexity in the turbulence modeling are suitable for vortex tube analysis.

On the base of the simulation studies of the vortex tube, made with the help of the program products Autodesk Inventor Professional and Autodesk Simulation CFD, an unstationary distribution of the temperature field and the velocity are obtained.

Further optimisation of the VT design is needed in order to improve its efficiency.

### Nomenclature

C – heat capacity, J/(kg.K);

$\rho$ – density, kg/m$^3$;

k – coefficient of heat transfer,W/(m.K);

t – temperature, °C;

### References

[1.] Ranque, G. J., *Experiences on Expansion in a Vortex with Simultaneous Exhaust of Hot Air and ColdAir (in French), J. Phys.Radium, 7 (1933), 4, pp. 112-114*

[2.] Hilsch, R., *The Use of Expansion of Gases in Centrifugal Field as a Cooling Process (in German), Z.Naturforschung, 1 (1946), pp. 208-214*

[3.] http://science.raderdev.com/

[4.] http://www.vortexitalia.net/

[5.] Aljuwayhel, N. F., Nellis, G. F., Klein, S. A., *Parametric and Internal Study of the Vortex Tube Using a CFD Model, Int. J. Refrig., 28 (2005), 3, pp. 442-450*

[6.] Skye, H. M., Nellis, G. F., Klein, S. A., *Comparison of CFD Analysis to Empirical Data in a Commercial Vortex Tube, Int. J. Refrig., 29 (2006), 1, pp. 71-80*

[7.] Bramo, A. R., et al., *Computational Fluid Dynamics Simulation of Length To Diameter Ratio Effects On The Energy Separation In A Vortex Tubethermal Science, Year 2011, Vol. 15, No. 3, pp. 833-848*

[8.] Hossein, N. A., Shamsoddini, R., *Numerical Three-Dimensional Analysis of the Mechanism of Flow and Heat Transfer in a Vortex Tube, Thermal Science, 13 (2009), 4, pp. 183-196*

[9.] Rahul B Patel, Dr. V. N. Bartaria, *Exprimental Investigation & Numerical Analysis Of Ranque Hilsch Vortex tube, International Journal of Engineering Research & Technology (IJERT) Vol. 2 Issue 1, January- 2013, pp. 1-6*