Applied Thermal Engineering 67 (2014) 494–506

Contents lists available at ScienceDirect

Applied Thermal Engineering

journal homepage: www.elsevier.com/locate/apthermeng

Investigation of the energy separation effect and flow mechanism inside a vortex tube

Xingwei Liu, Zhongliang Liu*

Beijing University of Technology, College of Environmental and Energy Engineering, 100124, China

HIGHLIGHTS

- Establish a 3-dimensional computational model for the given vortex tube.
- Five different turbulence models were tested for the numerical calculation.
- Experimental test facility was built to validate the computational model.
- Essence behaviors of the separation and strong swirling were displayed.

ARTICLE INFO

Article history:
Received 12 October 2013
Accepted 28 March 2014
Available online 5 April 2014

Keywords:
Vortex tube
Numerical investigation
Experimental validation
Flow parameters
Streamline contrast

ABSTRACT

Although many efforts have been made to disclose the energy separation phenomenon based on theoretical, numerical and experimental analysis in vortex tubes, it is still difficult to provide systematical information for designing vortex tubes due to the complexity of the physical process and the shortage of fully generally-applicable theory and methods. The purpose of current study was to search an effective method to predict the energy separation and flow behavior within a vortex tube. A three-dimensional computational fluid dynamic model together with the experimentally validated turbulence model is established and an experimental measurement is carried out using an optimal structure of vortex tube so as to validate the computational model. The flow characteristics including the total temperature and tangential velocity were obtained. The effects of cold fluid outlet diameter on the flow and temperature separation of a counter-flow vortex tube were investigated comprehensively. The streamlines with different cold fluid outlet diameters were presented and analyzed. A possible explanation for the cold fluid outlet diameter influences on the temperature difference between the cold fluid outlet and the hot fluid outlet were also given.

1. Introduction

The vortex tube also called Ranque–Hilsch vortex tube first observed by Ranque [1] is a remarkable simple thermal device capable of generating cold and hot fluid streams from a single source of compressed gas simultaneously. Vortex tubes are generally classified as counter-flow vortex tubes and parallel-flow vortex tubes according to the positioning of their cold fluid outlet. In counter-flow vortex tubes, the cold fluid outlet is placed on the opposite end of the hot fluid outlet. In parallel-flow tubes, the cold and hot fluid outlets are located at the same end. As different as the two configurations may look, the basic energy separation mechanisms are same. However, many researchers have proved that parallel-flow tubes perform less effective than equivalently proportional counter-flow designs. So in this paper the counter-flow geometry has been chosen. Shown in Fig.1, a vortex tube mainly consists of the following parts: one or more inlet nozzles, a vortex chamber, a concentric cold orifice located at one end of the tube and valve used for control the hot mass flow rate. Hilsch [2] methodically inspected the influences of the main factors including inlet pressure and geometrical parameters on the performance and his work is a comprehensive contribution to the possible explanation of the energy separation process. Westley [3] made a comprehensive review of the literature and achievements during 1931–1953 and provided some extended knowledge of the vortex tube. Sibulkin [4] proposed a theoretical model used for calculating the radial velocity and temperature distributions to explain the energy separation phenomenon inside the vortex tube.

---

* Corresponding author.
E-mail addresses: liuzhl@bjut.edu.cn, liuzl@bjut.edu.cn (Z. Liu).

http://dx.doi.org/10.1016/j.applthermaleng.2014.03.071
1359-4311/© 2014 Elsevier Ltd. All rights reserved.

Nomenclature

a	width of the inlet nozzle, mm
b	height of the inlet nozzle, mm
d_c	diameter of the cold fluid outlet, mm
e	internal energy
$\overrightarrow{g}$	gravitational acceleration, kg m s^{-2}
m	mass flow rate, kg/s
$\dot{m}_{in}$	total mass flow rate, kg/s
$\dot{m}_c$	cold mass flow rate, kg/s
z	the distance measured from the entrance location
k	turbulence kinetic energy, m^2/s^2
k_e	effective conductivity, W m^{-1} K^{-1}
$\bar{v}$	velocity components in x, y and z direction, m/s
L	total length of vortex tube, mm
Cp	specific heat at constant pressure, J kg^{-1} K^{-1}
E	volumetric total energy, J
$\overrightarrow{F}$	external body forces, N
G_b	generation of turbulence kinetic energy due to buoyancy , m^2/s^2
Y_M	contribution of the fluctuating dilatation
P	static pressure, Pa
$\dot{Q}_h$	hot energy separation, W
$\dot{Q}_c$	cold energy separation, W
R	gas constant, J/(mol K)
T	static temperature, K
I	unit tensor
D	diameter of the working tube, mm
Greek symbols	
μ	molecular viscosity, Pa s
ω	turbulence specific dissipation rate, 1/s
ε	turbulence dissipation rate, m^2/s^3
ρ	density, kg m^{-3}
$\bar{\bar{\tau}}$	stress tensor, N m^{-2}
Θ	dimensionless diameter of the cold fluid outlet
σ_ε	turbulent Prandtl numbers for ε
σ_k	turbulent Prandtl numbers for k
Γ_k	effective diffusivity of k
Γ_ε	effective diffusivity of ε
Ω_k	mean rate-of-rotation tensor
ξ	cold fluid mass fraction
ΔT	temperature difference between cold and hot fluid outlet, K
ΔT_c	temperature difference between cold fluid outlet and inlet, K
ΔT_h	temperature difference between hot fluid outlet and inlet, K
Subscripts	
c	cold fluid outlet gas
h	hot fluid outlet gas
cr	critical state
in	inlet gas

Due to its advantages of compactness, safety, without moving parts, and low cost, it has attracted a great attention from both scientists and industries from its first appearance and has been widely used in many fields, such as heating and cooling application, separation of gas mixtures, gas liquefaction, chemical industry, dehydration of natural gas, electric production etc. [5–8].

Owing to the attractive advantages of the vortex tube, many researchers have focused on this topic. For example, Linderstrøm-Lang [9] studied the transport of mass and energy in the vortex tube, and indicated the significant of the secondary flow as well as its interaction with the tangential velocity distribution. Saidi et al. [10] designed a reliable test rig to examine the effect of geometrical parameters such as diameter and length of main tube, diameter of cold fluid outlet orifice, shape of entrance nozzle. Gao et al. [11] carried out experiment study on the vortex tube. It can be concluded from his study that the secondary circulation flow exists in a vortex flow, and needs to be further explored. Nimbalkar et al. [12] conducted an experimental test using different cold end orifice diameters. Their results proved that cold fluid mass fraction is the crucial factor related to the performance of a vortex tube. More experimental investigations had been made by Eiamsa-ard [13], Dincer et al. [14–18], and Xue et al. [19,20], etc. However, as far as the present authors could know, all these experimental studies could not definitely explain the real mechanism of vortex tubes. It is well known that experimental tests are advantageous in predicting the overall performance and the influence of geometry. However, only a part of flow parameters can be collected by experiments, most of the detailed and important data are very difficult to obtain. These detailed data such as flow pattern and structure are very important for understanding the energy separation phenomenon.

Due to the limitation of the experimental study, people turn to computational fluid dynamics (CFD) to find numerical simulations for help to acquire the detailed flow characteristics inside the vortex tubes. Frohlingsdorf et al. [21] simulated the compressible flow and energy separation phenomenon with commercial software CFX. Aljuwayhel et al. [22] established a two-dimensional axisymmetric CFD model to investigate the vortex tube energy separation mechanism, and their results proved that their numerical model could predict the temperature separation successfully. Kazantseva et al. [23] investigated the swirling flow using a commercial software CFX-TASK. Skye et al. [24] adopted the same numerical method as Aljuwayhel [22] to demonstrate the performance analysis of the vortex tube and utilized the CFD model as a design tool. Behera et al. [25,26] developed a three-dimensional numerical model using the commercial CFD code (Star-CD) to analyze the flow characteristics and energy separation mechanism inside the vortex tube. Smith Eiamsa-ard et al. [27,28] carried out a numerical simulation to examine phenomena of the flow field and energy separation in the compressible vortex-tube flows. Pouraria et al. [29] studied on the performance of a vortex tube refrigerator with a divergent hot tube based on a two dimensional axis-symmetric swirl model. The above-mentioned work shows that the numerical simulation method can be used to reproduce the complicated flow process inside the vortex tubes, although some of the calculation models were conducted using two-dimensional model which is too simple to take all important factors into consideration. In order to obtain more accurate and detailed information, the 3D computational model has to be used.

Although many experimental, numerical and theoretical studies have been carried out so far and great efforts have been made to understand the energy separation phenomenon, the energy separation mechanism has been keeping unclear. The complexity of the flow inside the vortex tubes such as the existence of milt-circulation and eddy phenomenon makes the energy separation mechanism even more ambiguous. Therefore further clarification