International Journal of Heat and Mass Transfer 113 (2017) 432–443

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

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

Computational fluid dynamics simulation of heat transfer and fluid flow characteristics in a vortex tube by considering the various parameters

Abbas Moraveji, Davood Toghraie *

Department of Mechanical Engineering, Khomeinishahr Branch, Islamic Azad University, Khomeinishahr, Iran

ARTICLE INFO

Article history:
Received 26 March 2017
Received in revised form 17 May 2017
Accepted 24 May 2017

Keywords:
Numerical investigation
Temperature
Flow rate
Vortex tube

ABSTRACT

In this paper, the effects of number of inlets, tube length and diameter of cold outlet on temperature, flow rates passing through the vortex tube are investigated. The results including temperature of cold outlet and flow rates passing through the vortex tube are discussed. The effect of length, number of inlets, ranging from 1 to 5 inlets and the effect of cold outlet on the results are investigated. According to obtained results, we conclude that the passing flow rate from a cold outlet is increased as its diameter increase and by increasing the length of the vortex tube, the passing mass flow rates from the cold and hot cross-sections slightly increased and slightly increased, respectively. Also, the temperatures at both outlets decreased as the number of inlets increased, while increases were observed as the radius of cold outlet increased and the temperature of exiting gas is considerably higher than hot and cold outlets compared to the case where more number of inlets with reduced diameters is used. As shown, for $L/D = 15$ and as the radius of cold outlet is increased, the fraction of mass flow rate is decreased from 0.8 to 0.7 and then 0.6, from 0.65 to 0.58 and then 0.52, and from 0.42 to 0.32 and then 0.24 for $n = 1$, 3 and 5.

1. Introduction

Vortex tube is a simple device without a moving part which is capable of separating hot and cold gas streams from a higher pressure inlet gas stream. The vortex tube has been used in industrial applications of cooling and heating processes because of being a simple, compact, light and quiet. There are many places in natural gas industries where natural gas pressure needs to be reduced. This pressure reduction is currently accompanied with entropy generation or exergy destruction. For example, as a natural-gas distribution pipeline nears a customer, the high-pressure gas needs to be reduced to working level. Throttling valves are currently utilized to reduce the gas pressure. As natural gas pressure reduction takes places at throttling valves, the gas temperature is also lessened. The mechanism of energy separation has been investigated by several scientists and second law approach has emerged as an important tool for optimizing the vortex tube performance. So far there are very plenty of available researches to validate the reliability of CFD analyses for investigating the flow and temperature within the vortex tube.

Linhart et al. [1] studied vortex tube properties. Their work showed that the vortex tube flow processes is possible to simulate by means of accessible computation instruments, but it is necessary to modify the generally defined turbulent models.

Promvonge and Eiamsa-ard [2] investigated the vortex thermal separation in a vortex tube. Their results showed that the insulated vortex tube with 4 inlet nozzles and cold orifice diameter of 0.5D yielded the highest temperature reduction.

Eiamsa-ard and Promvonge [3] predicted vortex flow and thermal separation in a subsonic vortex tube. Their computations showed that temperature separation inside a vortex tube exists.

Eiamsa-ard and Promvonge [4] investigated the thermal separation in a Ranque–Hilsch vortex tube. They showed that the differences of results obtained from using the various schemes are marginal. In addition, results predicted by both turbulence models generally are in good agreement with measurements but the ASM performs better agreement between the numerical results and experimental data.

Kalal et al. [5] studied the vortex tube performance to investigate the parameters affecting vortex tube operation. Their results showed that for 1000 mm length vortex tube, the cold outlet diameter of 12 mm is ideal for the maximum temperature difference between hot and cold air temperature.

Eiamsa-ard and Promvonge [6] simulated the flow field and temperature separation in a vortex tube. Their results show that

* Corresponding author at: Department of Mechanical Engineering, Islamic Azad University, Khomeinishahr Branch, Khomeinishahr 84175-119, Iran.

E-mail address: Toghraee@iaukhsh.ac.ir (D. Toghraie).

http://dx.doi.org/10.1016/j.ijheatmasstransfer.2017.05.095
0017-9310/

Nomenclature

α	thermal diffusivity (m^2/s)	k	turbulent kinetic energy
μ	dynamic viscosity (kg/ms)	L	length of tube (mm)
ρ	density (kg/m^3)	m	mass (kg)
CI	mass flow rate from cold outlet to the total mass flow rate	n	number of inlet
C_p	specific heat (J/kg K)	Q	volumetric flow rate (m^3/h)
d	inlet diameter (m)	*Re*	Reynolds number
D	diameter (m)	T	temperature (K)
g	gravity acceleration (m/s^2)	u	velocity (m/s)
HC	mass flow rate from hot outlet to l mass flow rate cold outlet	ε	turbulence dissipation rate
HI	mass flow rate from hot outlet to the total mass flow rate	ν_t	kinematic eddy viscosity (m^2/s)
K	thermal conductivity (W/m K)	ν	kinematic viscosity (m^2/s)
		r	radius (m)

larger temperature gradients appear in the outer regions close to the tube wall for the static temperature contours and the separation effect or the difference of the total temperature is high in the core region near the inlet nozzle.

Secchiaroli et al. [7] simulated the turbulent flow in a Ranque–Hilsch vortex tube. Their results show significant differences in the velocity profiles, temperature profiles and secondary vortex structures, varying turbulence model.

Zin et al. [8] illustrated the influence of the length to diameter on the fluid flow characteristics inside the Ranque-Hilsch vortex tube predicted by computational fluid dynamic and validated through experiment. They simulated the swirl velocity, axial velocity, radial velocity component and also secondary circulation flow along with the pressure within the vortex tube.

Pourmahmoud et al. [9] simulated the effect of inlet gas temperature on the energy separation in a vortex tube. They found that increasing the inlet temperature does not have any significant effect on the stagnation point and maximum wall temperature position.

Rahbar et al. [10] simulated a micro scale Ranque-Hilsch vortex tube. Their results showed that fluid flow and energy separation inside the micro-scale vortex tube is quite similar to those of traditional ones.

Rafiee and Sadeghiazad [11] investigation on heat transfer and energy separation inside a counter flow vortex tube using different shapes of hot control valves. They found that the vortex tube with the truncated valve produces the highest cooling capability, also, the highest value of axial and tangential speeds belongs to this vortex tube and the vortex tube with the plate valve has the minimum axial and tangential velocities as well as the lowest cooling capability.

Rafiee and Sadeghiazad [12] investigated Efficiency evaluation of vortex tube cyclone separator. They showed that the cooling and heating abilities of the system will increase by 7.45 and 5.76% at the optimum flow fraction (respectively) as a result of increase in the convergence angle from 8 to 10.

Rafiee and Sadeghiazad [13] analyzed on the effect of rounding off edge radius on thermal separation inside a vortex tube. They determined the effect of changing radius of rounding off edge at hot tube entrance on vortex tube performance has been studied for different value of hot tube entrance and the optimized radius.

Rafiee and Sadeghiazad [14] investigated heat and mass transfer between cold and hot vortex cores inside ranque-hilsch vortex tube-optimization of hot tube length. They found that there is no need for the pressure values when the model is created based on the pressure-far-field boundary condition.

Rafiee and Sadeghiazad [15] analyzed the optimization of throttle angle for a convergent vortex tube. They found that the difference between the cold core and the flow field pressure creates a drag force that continuously affects how the fluid particles move towards the control valve.

Rafiee and Sadeghiazad [16] investigated on optimization of the air separator structural parameters for maximum separation efficiency. They explained the shape of flow domain inside the separator using three-dimensional computational fluid dynamics (CFD) models.

Pourmahmoud et al. [17] investigated numerical energy separation analysis on the commercial Ranque-Hilsch vortex tube on basis of application of different gases. They performed a comprehensive comparison is performed in this article between two different kinds of boundary conditions for cold and hot exhausts, i.e. pressure-outlet and pressure-far-field.

In this paper, we investigate the effects of number of inlets, tube length and diameter of cold outlet on temperature, flow rates passing through the vortex tube. Then, the results including temperature of cold outlet and flow rates passing through the vortex tube are discussed. The results comprise several subsections. In the first section, the effect of length on the results, including the ratios of output flow rate from hot outlet to the input flow rate as well as to the total input flow rate was investigated. Then, the effect of number of inlets, ranging from 1 to 5 inlets, on the considered results was investigated. The effect of cold outlet on the results is also included.

2. Geometry, material and governing equations

2.1. Geometry of the problem

The geometry of the vortex tube with $L/D = 15$ and $r_c = 0.007$ m and the meshing from different view are illustrated in Fig. 1.

2.2. Material

In the pervious works mentioned in the introduction section, air was used as the working fluid. The numerical study was conducted at this project showed that there are significant differences between the results of Air and Methane. Hence, in this simulation we use from Methane as the working fluid instead of Air and we consider it to be an ideal gas. Methane gas properties at zero degrees Celsius at atmospheric pressure have been shown in Table 1.