International Journal of Heat and Mass Transfer 78 (2014) 360–370

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

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

CFD analysis of energy separation of vortex tube employing different gases, turbulence models and discretisation schemes

H.R. Thakare *, A.D. Parekh

Sardar Vallabhbhai National Institute of Technology, Surat 395007, India

ARTICLE INFO

Article history:
Received 7 December 2013
Received in revised form 26 May 2014
Accepted 29 June 2014
Available online 23 July 2014

Keywords:
Vortex tube
Prandtl number
Thermal diffusivity
Specific gas constant
Thermal conductivity
Energy separation
Turbulence model
Secondary flow

ABSTRACT

This paper is aimed towards presenting the CFD study of vortex tube carried out to gain an understanding about influence of thermo-physical properties such as thermal diffusivity, Prandtl number, specific gas constant and thermal conductivity of different gases and turbulence models on its performance. The energy separation has been observed for eight different gases as working fluid. For the first time in CFD analysis of vortex tube, H_2 has been used as working fluid and its effect investigated. To understand the complex nature of highly compressible, turbulent and swirling flow within the vortex tube, different turbulent models, namely, one equation Spalart–Allmaras, two equations Standard k–ε and Standard k–ω model are used. Additionally, every turbulence model has been tested by using Second Order Upwind (SOU) and Quadratic Upstream Interpolation for Convective Kinetics (QUICK) scheme. Results indicated that magnitude of energy separation increases with increase in thermal diffusivity and thermal conductivity of the gas, with H_2 being an exception. H_2 also defies the criteria of decreasing molecular weight for improved energy separation. Magnitude of energy separation decreases with value of specific gas constant. Prandtl number of gas does not show any influence on energy separation magnitude. The energy separation magnitude predicted by using Spalart–Allmaras model in the cold region is better than that predicted by Standard k–ε and Standard k–ω model. Standard k–ε model combined with QUICK scheme presents itself as suitable model to predict the flow physics appropriately. A recirculating secondary flow has also been identified by all the models. The overall prediction of energy separation effect and flow physics parameters has been in good agreement with experimental results.

1. Introduction

In 193, Ranque [3] discovered the phenomenon of energy separation inside the vortex tube, when he found that tangentially admitted stream of compressed air was separated into two different streams of air simultaneously. One of the streams was colder while another one was hotter than inlet air stream. The admitted air stream undergoes expansion inside the nozzle, thus gaining high velocity and creating a vortex flow inside the tube. The hot stream exit is located on far side of the tube, while cold end is located near the tube inlet.

For many years, vortex tube has remained a topic of interest for scientific research due to this amazing temperature/energy separation effect, which happens in absence of any moving parts or chemical reaction inside the tube. The vortex tube is essentially simple in design, compact in size and light in weight. Due to its inherent advantages and features vortex tube is used in versatile applications of spot cooling.

Fulton [1] observed that kinetic energy transfer from core axial layers to peripheral fluid layers caused temperature rise of peripheral layers in vortex tube. Skye et al. [2] analysed energy separation using commercially available vortex tube and found that maximum power separation was obtained for cold mass fraction of about 0.65. His CFD model predicted that backflow of working fluid occurs at cold end at low cold mass fraction. Farouk et al. [4] carried out Large Eddy Simulation (LES) of vortex tube for the first time. He utilised the experimental results of Skye et al. and observed that energy separation predicted by LES was in better agreement with experimental results than those obtained using Standard k–ε model. Shamsoddini et al. [5] studied the effect of number of nozzles on flow structure and power of cooling of vortex tube. He observed that power of cooling increases with increase in nozzle numbers. Eiamsa-ard et al. [6] used Algebraic Stress Model [ASM] and Standard k–ε turbulence model for simulation of thermal energy separation and found that ASM had better agreement with experimental results. Promvonge et al. [8] performed

---

* Corresponding author.
E-mail address: thakare.hitesh@gmail.com (H.R. Thakare).

http://dx.doi.org/10.1016/j.ijheatmasstransfer.2014.06.083
0017-9310/© 2014 Elsevier Ltd. All rights reserved.

Nomenclature

C_p	specific heat at constant pressure, J kg^{-1} K^{-1}
K	Kelvin
k	thermal conductivity, W m^{-1} K^{-1}
k_{eff}	effective thermal conductivity, W m^{-1} K^{-1}
M	molecular weight, kg $kmol^{-1}$
Pr	Prandtl number
QUICK	Quadratic Upstream Interpolation for Convective Kinetics
R	specific gas constant, J kg^{-1} K^{-1}
r	radius of vortex tube, m
SOU	Second Order Upwind
T	static temperature
T_c	cold region temperature
T_0	total temperature or stagnation temperature
ΔT_c	cold region temperature separation
V	velocity
x	axial distance from left end of the vortex tube, mm

Greek symbols

ρ	density (kg m^{-3})
α	thermal diffusivity, (m^2 s^{-1})
μ	viscosity (kg m^{-1} s^{-1})

Subscripts

c	cold region
cfd	CFD result
exp	experimental result
t	turbulent

combined study on the effect of number of inlet nozzles, cold end diameter, and tube insulations on the temperatures eparation inside the vortex tube and observed that temperature separation increased with increase in number of inlet nozzles. Khazaei et al. [18] used different gases in vortex tube and concluded that magnitude of energy separation depends upon molecular weight of the gas and ratio of specific heats of gas. Soni [9] performed analysis using 170 different vortex tubes and found that for good performance, ratio of area of cold end to area of vortex tube should be in the range of 0.08 to 0.145. Pourmahmoud et al. [12] observed that helical nozzles are more suitable to achieve higher magnitude of energy separation as compared to straight nozzles. Behera et al. [13] optimized the dimensions of vortex tube using CFD and found that secondary circulation was eliminated for ratio of cold end diameter to vortex tube diameter equal to 0.58. Akhesmeh et al. [15] successfully predicted the velocity profiles inside vortex tube using CFD analysis. The results of CFD analysis were in good agreement with experimental results. Yilmaz et al. [16] presented an extensive and useful review of published literature on various investigations of vortex tube. Ons TLILI EL MAY et al. [17] used two different turbulence models during CFD analysis and achieved good agreement with experimental results. Ahlborn et al. [11] predicted the presence of secondary circulation flow inside the vortex tube for smaller cold end diameter. Frohlingsdorf et al. [19] used code system CFX for analysis and found that influence of unsteady effects can be incorporated in the numerical model by increasing the turbulent Prandtl number. Stephan et al. [22] carried out experimental study on vortex tube using air as working fluid and found that Gortler vortex produced by tangential velocity on inside wall of vortex tube was a major driving force for energy separation. Aydin et al. [23] investigated various design parameters of vortex tube and observed that inlet pressure and cold mass fraction are important parameters which significantly affect the performance of vortex tube. Farouk et al. [24] used LES to simulate the separation of gas species for a nitrogen - helium gas mixture. Authors found that over the entire cold mass fraction range, very small amount of gas separation was predicted. Saidi et al. [25] performed experimental investigation on vortex tube and found that presence of moisture in admitted air negatively affected the cold end temperature separation.

Despite being simple in design and working, the detailed and universally accepted explanation about mechanism of energy separation is yet to be put forth. Experimental setup appears to be facing significant difficulties to predict the flow physics inside the vortex tube [7]. Here, CFD study may help the researchers significantly.

1.1. Objectives of present study

Present numerical analysis has been carried out on vortex tube of Hartnett et al. [21]. It is depicted in Fig. 1. The diameter of vortex tube used is 0.0762 m and length of tube is 0.77 m. The experimental study of [21] used vortex tube with cold end being completely closed and air as working fluid. In the original experimental investigation, air was admitted into the vortex tube through 8 nozzles placed on circumference of the tube at pressure of 2.3 atm (abs).

First objective of this study is to obtain understanding about flow fields and energy separation phenomenon inside the vortex tube when different gases have been used as working fluid; instead of air only. For the first time in numerical analysis of uniflow vortex tube, hydrogen has been used as a working fluid to observe its energy separation. The lower cost of CFD simulations as compared to experimental arrangement makes such parametric study possible and feasible. Practically, H_2 finds important application as cryogenic propellant. In total, 8 gases have been tested for simulation, namely, He, N_2, O_2, Methane, water vapour, air, CO_2, and hydrogen. Out of these, 7 gases have already been numerically studied by Khazaei et al. [18], except hydrogen. However, Khazaei et al. [18] has analysed the energy separation effect only with regards to molecular weight and ratio of specific heats of gas. Also, Khazaei et al. [18] did not discuss the effect of discretisation schemes on

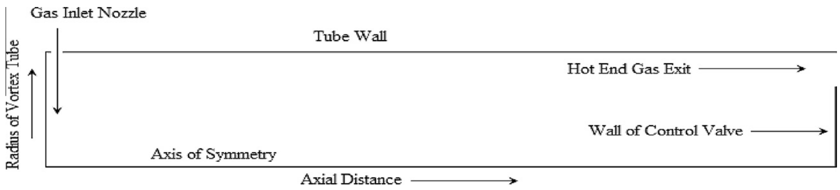

Fig. 1. Schematic of computational domain for vortex tube of Hartnett et al. [21].