Energy 85 (2015) 62–77

Contents lists available at ScienceDirect

Energy

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

Computational analysis of energy separation in counter—flow vortex tube

Hitesh R. Thakare*, A.D. Parekh

Department of Mechanical Engineering, Sardar Vallabhbhai National Institute of Technology, Surat, Gujarat, 395007, India

ARTICLE INFO

Article history:
Received 24 November 2014
Received in revised form
8 February 2015
Accepted 5 March 2015
Available online 18 April 2015

Keywords:
Counter flow vortex tube
Turbulence model
Swirl velocity
Temperature separation

ABSTRACT

Present paper is aimed towards reporting CFD study carried out on counter—flow vortex tube using different gases at various values of cold mass fraction and using different turbulence models. In CFD analysis of counter flow vortex tube, various working gases have been seldom used and their energy separation effect studied relative to cold mass fraction. Also, computational efforts to compare results of one equation Spalart Allmaras model used for analysis of counter—flow vortex tube with other two equation turbulence models, that is, Standard k-ε and Standard k-ω model as well as RSM (Reynolds Stress Model) have been seldom reported.

All turbulence models are observed to predict similar flow physics inside vortex tube, however, with different magnitude. Spalart Allmaras model over predicts while RSM under predicts temperature separation magnitude. Nitrogen as working fluid of vortex tube produces highest temperature separation, while it is least for CO_2 among the working fluids studied. Cold mass fraction is an important parameter which directly affects the temperature separation. Cooling power separation should also be considered as an important performance parameter of vortex tube, instead of cold end temperature alone.

Results of present CFD study are in better agreement with experimental results than previous CFD results.

1. Introduction

A vortex tube is a simple and compact device capable of producing simultaneous streams of hot and cold gas, when compressed gas is admitted into it in tangentially direction. The inlet stream of compressed gas undergoes expansion in the inlet nozzles so that it attains higher tangential velocity. This causes a vortex type of flow to be set up inside the tube. Due to internal mechanism, temperature of one stream coming out of vortex tube is less, while that of another stream is higher than that of inlet pressurised gas. This phenomenon is known as energy separation or temperature separation. The mass of hot and cold gas coming out of vortex tube can be controlled with the help of control valve located on hot side. The magnitude of temperature separation changes with the change in mass of cold gas extracted i.e. cold mass fraction, the pressure of inlet stream and geometry of vortex tube.

This energy separation or temperature separation effect is produced instantaneously without any moving parts inside the tube or any chemical reaction. Absence of moving parts results in very low maintenance and hence longer service life of the tube. Ability of vortex tube to produce instantaneous hot and cold air makes it suitable for applications such as spot cooling of electronic components, cooling suits for mine workers and spray painters, thermal sensors testing, liquefaction of gases, separating particles in waste gas industry etc.

Ever since its discovery made by Ranque [1], vortex tube has remained a topic of significant interest for researchers. Many researchers have conducted experimental studies to predict performance of vortex tube for various set of parameters, without reaching any universal agreement. The results of experimental investigation are mostly limited to average or integral values owing to larger pressure gradients within smaller dimensional domain of vortex tube, which is further complicated due to the presence of very high velocity swirling and turbulent flow inside the tube. As an example, Ahlborn and Groves [2] reported that even use of a 1.6 mm diameter Pitot tube probe for a vortex tube of 25.4 mm diameter resulted in blockage of 8% of vortex tube cross section. At

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

http://dx.doi.org/10.1016/j.energy.2015.03.058
0360-5442/

Nomenclature

Cp	specific heat at constant pressure, J kg^{-1} K^{-1}
K	Kelvin
k	thermal conductivity, W m^{-1} K^{-1}
L	total length of vortex tube, mm
M	molecular weight, kg $kmol^{-1}$
m	mass flow rate of fluid, kg s^{-1}
$\dot{Q}_C$	cooling power separation, kW
T_c	cold end temperature, K
ΔT_c	cold end temperature separation, K
T_h	hot end temperature, K
T_o	total temperature, K
T_s	static temperature, K
V	velocity, ms^{-1}
x	axial distance from left end of the vortex tube, mm

Greek symbols

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

Subscripts

c	cold region value
cfd	CFD result
exp	experimental result

this point of uncertainty, CFD can be used to obtain detailed profile of flow physics parameters of swirling and turbulent flow inside the vortex tube. It is a fact that CFD studies cannot act as a replacement for experimental studies. However, CFD studies can perform very well as complimentary to experimental studies, if carried out thoroughly and systematically. Also, the time required and the cost of CFD study is much less than that of experimental ones, if appropriate CFD model could be established. This facilitates a number of parametric studies to be carried out. Secchiaroli et al. [3] used RSM (Reynolds Stress Model) successfully for the first time in 2D axisymmetric computational model of vortex tube. Authors also used LES (Large Eddy Simulation) and RNG k-ε turbulence models for analysis and concluded that velocity field prediction by RSM was better than other two models, regardless of computational time required. This is contrary to the findings of Skye et al. [4] who reported a declined accuracy of CFD data relative to experimental data, when RNG k-ε turbulence model was used. Skye et al. [4] also reported that they could not make Reynolds stress equations to converge during their simulation. Skye et al. [4] performed experimental and CFD study on commercial vortex tube. Authors developed 2 Dimensional CFD model of vortex tube using 25,000 cells and observed that their CFD model under predicted the temperature separation at cold and hot end than experimental results. Farouk and Farouk [5] used LES (Large Eddy Simulation) for the first time in the CFD analysis of vortex tube and identified the presence of low swirl velocity zone in core axial region of tube, which coincided with zone of lowest temperature. For this, authors used experimental results of Skye et al. [4] and developed 2D axisymmetric computational model of vortex tube. The LES model also predicted the presence of secondary vortices in different regions of tube. Shamsoddini et al. [6] observed that axisymmetric CFD model was very much capable to predict the flow structure analogous to vortex tube having multiple inlet nozzles. Cockerill [7] reported that measuring probes can influence the flow pattern inside vortex tube which creates difficulty in experimental determination of velocity components of fluid inside vortex tube. CFD Analysis of *uni—flow* vortex tube carried out by Khazaei et al. [8] using 2D computational domain of vortex tube and employing different gases revealed that maximum temperature separation was observed for Helium as working gas. Experimental performance study conducted by Aydin and Baki [9] using Air, Oxygen and Nitrogen as working gas for vortex tube revealed that highest temperature difference was obtained for Nitrogen. Dutta et al. [10] performed CFD analysis on 2D axisymmetric model of vortex tube consisting of 40,000 cells using various turbulence models namely, Standard k-ε, RNG k-ε, Standard k-ω and SST k-ω. Authors concluded that hot and cold end temperature separation predicted by Standard k-ε, Standard k-ω and SST k-ω model were reasonably closer to experimental results and found suitable for practical design purposes, while results of RNG k-ε deviated from experimental results. Baghdad et al. [11] performed CFD analysis of 3D computational domain using various turbulence models namely, Standard k-ε, Standard k-ω, SST k-ω and RSM. Authors concluded that all the turbulence models were capable to replicate general fluid dynamics and energy separation within vortex tube, however, with varying magnitude.

1.1. Objectives of present study

For the present CFD study, experimental results from study of Skye et al. [4] have been utilized. Skye et al. [4] also carried out CFD study on an axisymmetric model of vortex tube using 25,000 cells. During the CFD study of Skye et al. [4], even though qualitative agreement was achieved in terms of trends of temperature separation, however, CFD model under—predicted the temperature separation at cold and hot end. Hence, first objective of this study is to establish a reliable 2 Dimensional axisymmetric CFD model. To validate the reliability of CFD model, results of present CFD simulation are compared with original experimental and CFD results of Skye et al. [4] and also with CFD results of Farouk and Farouk [5] and Pourmahmoud and Akhesmeh [17], which also used 2 Dimensional axisymmetric model.

Once a reliable CFD model has been built, second objective of study is to observe the magnitude of the temperature separation effect produced for.

a) Various values of cold mass fraction (i.e. ratio of mass of cold gas exiting the tube to mass of admitted gas),
b) Different Turbulence Models
c) Different Working Gases

Modelling the flow physics and energy separation phenomenon inside the vortex tube is a challenging task due to the presence of highly turbulent, compressible flow accompanied by highly swirling motion of fluid inside the tube. To simulate the flow and analyse the fluid dynamics parameters correctly, it is necessary to employ an appropriate turbulence model. Study of Farouk and Farouk [5] indicated that LES results agreed better with experimental results. Similar observation was reported by Eiamsa-ard and Promvonge [12] during their study on 2D axisymmetric domain of *uni flow* vortex tube using ASM (Algebraic Stress Model). Also, DNS (Direct Numerical Simulation) approach is the best method to fully resolve the turbulence associated with the flow inside the vortex tube because it solves all the time and spatial scales in the velocity field. However, DNS approach is highly expensive in terms of computational cost. The computational efforts required for LES and ASM are also enormous [10]. The study of Secchiaroli et al. [3] reported a CPU time of 26 days by using LES, as compared to 1.5–2 h in case of RANS models. Pertaining to these observations, it is necessary to obtain a balance between computational cost and prediction accuracy delivered by turbulence model. For this