International Journal of Thermal Sciences 50 (2011) 2377–2385

Contents lists available at ScienceDirect

International Journal of Thermal Sciences

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

Numerical study of energy separation in a vortex tube with different RANS models

Mohammed Baghdad [a], Ahmed Ouadha [a,*], Omar Imine [a], Yacine Addad [b]

[a] *Laboratoire d'Energie et Propulsion Navale, Faculté de Génie Mécanique, Université des Sciences et de la Technologie d'Oran Mohamed BOUDIAF, B.P. 1505, Oran El-Mnouar, 31000 Oran, Algeria*

[b] *Nuclear Engineering Faculty, Khalifa University, P.O. Box 127788, Abu Dhabi, United Arab Emirates*

ARTICLE INFO

Article history:
Received 10 January 2011
Received in revised form
20 July 2011
Accepted 21 July 2011
Available online 24 August 2011

Keywords:
Vortex tube
Numerical study
Energy separation
RANS models

ABSTRACT

The aim of the present paper is to investigate numerically the energy separation mechanism and flow phenomena within a vortex tube. A 3D computational domain has been generated considering the quarter of the geometry and assuming periodicity in the Azimuthal direction which was found to exhibit correctly the general behaviour expected from a vortex tube. Air is selected as the working fluid. The flow predictions reported here are based upon four turbulence models, namely, the $k-\varepsilon$, $k-\omega$ and *SST* $k-\omega$ two-equations models and the second moment closure model (*RSM*). The models results are compared to experimental data obtained from the literature. Four cases have been considered by changing the inlet pressure from 200 up to 380 kPa. It has been observed that all the above mentioned models are capable of predicting fairly well the general flow features but only the advanced *RSM* model is capable of matching correctly the measured cold and hot outlet temperatures. All the other models over predict the mean temperature difference by values up to twice the measured data.

1. Introduction

Vortex tube is a simple energy separating mechanical device that produces hot and cold streams simultaneously from a compressed gas source. It is compact and simple to fabricate with no moving parts. This device can be used for cooling and heating. Nowadays, vortex tubes provide attractive applications, such as gas dehydration, removal of water vapour and droplets from natural gas streams, gas dewpointing, and process and spot cooling for industrial processes with cold air guns.

Since its invention by Ranque in 1933 and its improvement by Hilsch [1], many efforts have been made to explain the thermal separation phenomenon based on theoretical, numerical and experimental analysis. However, its coefficient of performance remained low and the mechanism of energy separation still ambiguous. Since then, other proposals have followed along the years in order to achieve performance enhancements by; increasing the number of inlet nozzles, changing the tube diameter, varying the cold and hot outlet diameters, and/or changing the working fluid [2–9].

Other studies were focussed in understanding the global behaviour of the flow in vortex tube by means of thermodynamic laws to determine the optimal performances of this device [10–12].

Nowadays, new development in CFD tools and measurement techniques can help understand better some of the flow phenomena and allow for the visualisation of the fluid flow paths within the vortex tube which would then on its turn, allow investigating how this energy separation phenomenon is occurring. Consequently, in the last decade, research was focussed in producing detailed data on the flow in different vortex tube configurations by both; experimental measurements and numerical predictions. For instance, the experimental work was mainly concentrated on the influence of the geometry and the parameters of the vortex tube on its performance [13–19].

In parallel, several numerical studies have been dedicated to this subject. For example, Fröhlingsdorf and Unger [20] simulated numerically the compressible flow and energy separation phenomena using the CFD code *CFX*. They extended an axisymmetric model by integrating relevant terms for the shear-stress-induced mechanical work. Behera et al. [21] conducted numerically a detailed parameters analysis of a vortex tube. The velocity components and the flow patterns have been evaluated using the CFD code *Star-CD*. Optimal design parameters of the vortex tube, such as number of nozzles, nozzle profiles, cold end diameter, length to diameter ratio and cold and hot gas fractions, have been also determined. Their paper also showed comparison between the CFD predictions and experimental measurements. Aljuwayhel et al. [22] investigated numerically the energy separation mechanism and flow phenomena within a counter-flow vortex tube. A two-dimensional axisymmetric computational domain was used for

* Corresponding author. Tel.: +213 6 61204325.
E-mail address: ouadha@univ-usto.dz (A. Ouadha).

1290-0729/$ – see front matter © 2011 Elsevier Masson SAS. All rights reserved.
doi:10.1016/j.ijthermalsci.2011.07.011