International Communications in Heat and Mass Transfer 35 (2008) 937–947

Contents lists available at ScienceDirect

International Communications in Heat and Mass Transfer

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

Numerical simulation of flow field and temperature separation in a vortex tube ☆

Smith Eiamsa-ard [a,*], Pongjet Promvonge [b]

[a] Department of Mechanical Engineering, Faculty of Engineering, Mahanakorn University of Technology, Bangkok 10530, Thailand
[b] Department of Mechanical Engineering, Faculty of Engineering, King Mongkut's Institute of Technology Ladkrabang, Bangkok 10520, Thailand

ARTICLE INFO

Available online 23 May 2008

Keywords:
Ranque–Hilsch vortex tube
Vortex flow
Temperature separation
Energy separation

ABSTRACT

A numerical analysis of flow field and temperature separation in a uni-flow vortex tube type is described. Effects of the turbulence modeling (k–ε model and ASM), numerical scheme (hybrid, upwind and second-order upwind) and grid density on calculation of energy separation in the vortex tube are also conducted. It is found that the calculated results are in reasonably good agreement with the experimental data for both the static and total temperatures; the use of the ASM improves slightly the accuracy of the predictions than that the k–ε model. It is also observed that larger temperature gradients appear in the outer regions close to the tube wall for the static temperatures and the separation effect or the difference of the total temperature is high in the core region near the inlet nozzle. The maximum total temperature in the field is visible at the axis location of x/D_o between 0.5 and 1.0 for the ASM.

1. Introduction

Vortex flows can be found in nature, such as tornadoes, and have been of considerable interest over the past decades due to their applications in many engineering applications for example: heat exchanger [1–4], vortex combustor [5–6] and vortex tube. Ranque–Hilsch vortex tube (or vortex tube) is a device enabling producing the hot and cold air when the compressed air flows tangentially into the vortex chamber through the inlet nozzles. This causes the vortex and swirl flow movement inside the vortex tube. The air in the middle region of the tube has a lower velocity and lower temperature than the inlet air on the other hand the air near the wall tube has higher velocity and higher temperature than the inlet air which is referred to as the temperature (or energy) separation effect. Nowadays the property of vortex tube has a great variety of application in industry which operating as a refrigerating machine. It has been widely used in the cooling industrial fields especially grilling, turning and welding on account of the various advantages of vortex tube such as cooling without moving part, non-electricity consuming, tiny, lightweight and inexpensive working chemical substance inside the vortex tube, uncomplicated cooling point, cleanliness, convenience and non-CFC's free from pollution. In general, vortex tube has been known in different names. The most well known names are: vortex tube, Ranque vortex tube, Hilsch vortex tube or Ranque–Hilsch, and Maxwell–Demon vortex tube. Through there are various names, "Ranque–Hilsch vortex tube" and "vortex tube" have been used in this present paper.

A literature review has demonstrated the existence of an extensive research regarding the energy separation in the vortex tubes. The vortex tube was first observed by Ranque [7,8]. Interest in the device was revived by Hilsch [9], who reported an account of his own comprehensive experimental and theoretical studies aimed at improving the efficiency of the vortex tube. The energy separation was first explained by Fulton [10]. He hypothesized that the inner layers of the vortex expand and grow cold while they press upon the outer layers to heat the latter. Sibulkin [11] replaced the steady three-dimensional flow problem by an unsteady, two-dimensional problem by replacing the axial coordinate with time. Reynolds [12] performed numerical analysis of a vortex tube. A detailed order-of-magnitude analysis was used for the various fluxes appearing in the turbulent energy equation and the prediction was compared with his measurements. Lewellen [13] combined the three Navier–Stokes equations for an incompressible fluid in a strong rotating axisymmetric flow with a radial sink flow and arrived at an asymptotic series solution. Linderstrom-Lang [14] examined analytically the velocity and thermal fields in the tube. He calculated the axial and radial gradients of the tangential velocity profile from prescribed secondary flow functions on the basis of a zero-order approximation to the momentum equations developed by Lewellen [15] for an incompressible flow.

Schlenz [16] investigated numerically the flow field and the process of energy separation in a uni-flow vortex tube. Calculations were carried out assuming a 2D axisymmetric compressible flow and using the Galerkin's approach with a zero-equation turbulence model to solve the mass, momentum, and energy conservation equations to calculate the flow and thermal fields. A numerical study of a large counter-flow vortex tube with short length was conducted by Amitani et al. [17]. The mass, momentum and energy conservation equations in a 2D flow model with an assumption of a helical motion in the axial

☆ Communicated by W.J. Minkowycz.

* Corresponding author.
E-mail address: smith@mut.ac.th (S. Eiamsa-ard).

0735-1933/$ – see front matter © 2008 Elsevier Ltd. All rights reserved.
doi:10.1016/j.icheatmasstransfer.2008.04.010

direction for an invicid compressible perfect fluids were solved numerically. Stephan et al. [18] formulated a general mathematical expression for the energy separation process but this could not be solved because of the complicated system of equations. Flohlingsdorf and Unger [19] studied on the phenomena of velocity and energy separation inside the vortex tube through the code system CFX with the k–ε model (first-order turbulence model). Behera et al. [20] investigated the effect of the different types of nozzle profiles and number of nozzles on temperature separation in the counter flow vortex tube using the code system of Star-CD with 'Renormalization Group' (RNG) version of the k–ε model. Aljuwayhel et al. [21] reported the energy separation and flow phenomena in a counter-flow vortex tube using the commercial CFD code FLUENT and found that the RNG k–ε model performed better than the standard k–ε model. This is contrary to results of Skye et al. [22] claimed that for vortex tube's performance, the standard k–ε model is better than the RNG k–ε model despite using the same commercial CFD code FLUENT. The simulation of thermal separation in a Ranque–Hilsch vortex tube was also reported by Eiamsa-ard and Promvonge [23–25] using the standard k–ε model and an algebraic Reynolds stress model (ASM). Their computations showed that results predicted by both turbulence models generally were in good agreement with measurements but the ASM performs better. T. Farouk and B. Farouk [26] introduced the large eddy simulation (LES) technique to predict the flow fields and the associated temperature separation within a counter-flow vortex tube for several cold mass fractions. However, most of the computations found in the literature used simple or the first-order turbulence models which are considered unsuitable for complex, compressible vortex tube flows. The present work presents a two-dimensional numerical investigation of flow and temperature separation behaviors inside a uni-flow vortex tube. The influence of several grid densities, numerical schemes, and turbulence modeling are also investigated. Comparisons of the calculated gas velocity, pressure and temperature with the measurements of a uni-flow vortex tube [27,28] are made to evaluate the turbulence models used.

2. Methodological and numerical considerations

2.1. Governing equations

For steady, compressible flows the Favre-averaged mean equations of motion, the turbulence kinetic energy (TKE) equation, the energy equation and the equation of state in Cartesian tensor notation can be summarised as:

$$\frac{\partial}{\partial x_i}(\bar{\rho}\tilde{u}_i)=0 \tag{1}$$

$$\frac{\partial(\bar{\rho}\tilde{u}_i\tilde{u}_j)}{\partial x_j}=-\frac{\partial\bar{p}}{\partial x_i}+\frac{\partial}{\partial x_j}(\bar{t}_{ij}+\tau_{ij}) \tag{2}$$

$$\frac{\partial}{\partial x_j}(\bar{\rho}\tilde{u}_j k)=\tau_{ij}\frac{\partial\tilde{u}_i}{\partial x_j}-\bar{\rho}\varepsilon+\frac{\partial}{\partial x_j}\left(\overline{t_{ji}u_i''}-\bar{\rho}\frac{1}{2}\widetilde{u_j''u_i''u_i''}-\overline{p'u_j''}\right)-\overline{u_i''}\frac{\partial\bar{p}}{\partial x_i}+\overline{p'\frac{\partial u_i''}{\partial x_j}} \tag{3}$$

$$\frac{\partial}{\partial x_j}\left(\bar{\rho}\tilde{u}_j\tilde{E}\right)=\frac{\partial}{\partial x_j}\left(\overline{t_{ji}u_i''}-\overline{p'u_j''}-\bar{\rho}\frac{1}{2}\widetilde{u_j''u_i''u_i''}\right)-\frac{\partial}{\partial x_j}\left(\overline{q_{L_j}}+\bar{\rho}\widetilde{u_j''h''}\right)+\frac{\partial}{\partial x_j}(\tilde{u}_i(\bar{t}_{ij}+\tau_{ij}))-\frac{\partial}{\partial x_j}(\tilde{u}_j\bar{p}) \tag{4}$$

$$\bar{p}=\bar{\rho}R\tilde{T}=(\gamma-1)\left(\bar{\rho}\tilde{E}-\bar{\rho}\tilde{u}_i\tilde{u}_i-\bar{\rho}k\right) \tag{5}$$

In the preceding, an overbar indicates the mean relative to Reynolds averaging, with a single prime for fluctuation. A tilde and a double prime are corresponding ones for Favre averaging. Also, x_i are the coordinate directions, and ρ is density, u_i are the velocities in the three coordinates directions, k defined by $\bar{\rho}k=\frac{1}{2}\overline{\rho u_i''u_i''}=\frac{1}{2}\bar{\rho}\widetilde{u_i''u_i''}$ is the turbulence kinetic energy, $\bar{p}$ is mean pressure, $\tilde{E}\equiv C_v\tilde{T}+\frac{1}{2}\tilde{u}_i\tilde{u}_i+\frac{1}{2}\widetilde{u_i''u_i''}$ is the mean total energy, and γ is the ratio of specific heats (C_p/C_v). $\bar{q}_L$ is the mean heat flux and the mean viscous stress tensor is approximated as:

$$\bar{t}_{ij}=\bar{\mu}\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)-\frac{2}{3}\bar{\mu}\frac{\partial u_k}{\partial x_k}\delta_{ij} \tag{6}$$

Finally, $\tau_{ij}=-\overline{\rho u_i''u_j''}=-\bar{\rho}\widetilde{u_i''u_j''}$ is the Favre-averaged Reynolds stress tensor.

The mean conservation equations have resulted in additional terms: τ_{ij}, $\overline{t_{ij}u_j''}$, $\overline{p'u_i''}$, $\bar{\rho}\widetilde{u_i''h''}$ and $\bar{\rho}\widetilde{u_j''u_i''u_i''}/2$ that require modelling. The modelling of some of the unclosed terms in these equations is based on their incompressible models whereas explicit compressible models are required for others. In the present study, two turbulence closure models are used, namely the standard k–ε model and an algebraic Reynolds stress model (ASM). The more detail of each model can be found in ref. [24].

2.2. Vortex tube of Lay

Measurements of Lay [27,28] were made in a Lucite tube having an inside diameter of D_o=50.8 mm and a length of l=1,066.8 mm. Air at 25.6 °C entered the tube tangentially through a single nozzle of d_n= 9.525 mm diameter and left the tube through a cone-shaped valve. Experimental velocity, pressure and temperature profiles were provided at 6 axial stations, namely; x=50.8, 228.6, 406.4, 584.2, 762 and 939.8 mm (or x/D_o=1.0, 4.5, 8.0, 11.5, 15.0 and 18.5 respectively) from the inlet nozzle, with the nozzle supply pressure (p_o) at about 1.68 atm (abs.). The total velocity was measured with a hot-wire anemometer. The tangential velocity profiles and the rate of mass flow including detail of the cone-shaped valve were not provided.

2.3. Boundary conditions

The calculated mean quantities are compared with available measurements at selected stations. Basic assumptions for all the computations of the particular vortex tube flows are made as follows: 2D axisymmetrical, subsonic flow inside the vortex tube, uniform flow properties at the inlet and ideal gas. Since the system is assumed to be an axisymmetric flow, only half of the flow domain needs to be considered throughout and special treatment for the flow at the inlet must be made for the computations. At the inlet, a circumferential slot is assumed instead of the actual inlet nozzles. For simplicity in the present computation the cone-shaped valve used as a discharge valve at the exit is replaced with a block valve.

2.4. Computational domain

The computational domains of the flow system are shown in Figs. 1 and 2A, while Fig. 2B illustrates a 60×25 non-uniform grid density used. Because detail of the cone-shaped valve used at the exit was not available, an assumed opening of the valve is used instead. Preliminary test in the present study showed that an opening of the discharge valve in a range of 0.5R and R did not affect much the flow and temperature fields in this tube. The speed of sound c, and the Reynolds number at the inlet calculated from local measured data are about 205 m/s and 694,000 respectively. Therefore the inlet Mach number, based on the relation $M_{in}=V_n/c$, is 0.6, indicating that the flow in the tube is compressible.

2.5. Numerical method

In the present computation the Favre-averaged Navier–Stokes equations, Eqs. (1) and (2); the TKE equation, Eq. (3); the energy