Numerical simulation for three-dimensional flow in a vortex tube with different turbulence models

Zhuohuan Hu, Rui Li, Xin Yang, Mo Yang, and Yuwen Zhang

School of Energy and Power Engineering, University of Shanghai for Science and Technology, P.R. China; Department of Mechanical and Aerospace Engineering, University of Missouri, Columbia, USA

ABSTRACT
Three-dimensional flow in a vortex tube with straight nozzle was investigated numerically. Four different turbulence models, Spalart-Allmaras, standard $k-\varepsilon$, SST $k-\omega$, and realizable $k-\varepsilon$ were proposed to replicate the flow in the vortex tube. The simulation results of pressure, temperature and velocity field were compared with the experimental data based on dimensionless analysis. It was found that the sudden gas expansion in the vortex chamber causes a pressure drop. The secondary circulations phenomena were observed and identified. Furthermore, the friction between the gas and wall surface and the energy transfer between different laminar flows are the main factors of energy separation in the vortex tube. The realizable $k-\varepsilon$ model predicts energy separation better than other models studied in this research.

1. Introduction

Researches on vortex tubes started from discovery of the “vortex tube effect” by Ranque, a French metallurgist, in the 1930s [1]. Later, a German physicist R. Hilsch made a significant contribution to the research of vortex tube [2]. The vortex tube was also named Ranque-Hilsch vortex tube in honor of the two founding scientists. The structure of the vortex tube generally includes one or more nozzles, vortex chamber, cold orifice, hot end tube, and a regulating valve. The compressed gas is tangentially injected into the vortex chamber through the nozzles. Part of the gas exit from the cold orifice with a temperature lower than the inlet temperature. The rest of gas escapes from the hot end tube at a higher temperature. The mass ratio of the cold air flow to heated air flow can be adjusted by the regulating valve. The airflow in a counter-flow vortex tube is illustrated in Figure 1.

Research on vortex tube has been carried out for several decades, but the principle of vortex tube effect is still puzzling. Relevant theories about the pressure gradient [3], viscous friction [4], secondary flow [5] and acoustic streaming have been developed [6]. However, there is no complete explanation for energy separation. The accurate descriptions of flow field and temperature field are crucial to revealing the physical mechanism of the vortex tube. It must be admitted that the small volume and the strong swirl flow of vortex tube have made the challenge to the field measurement. Gao et al. [7] developed a Pitot tube in order to mitigate the impact of measuring the flow field. The overall view of the experimental setup is presented in Figure 2. Xue et al. [8, 9] studied the flow field inside tube by visual measurement and demonstrated the existence of...
multi-circulations. In addition, the 2-D laser Doppler velocimetry was adopted by Guo et al. [10] to investigate the axial velocity distributions of the flow. And the variation of the reverse flow boundary was also discussed.

Numerical simulation can overcome the difficulty of measurement and it has been proved to be applicable to study the turbulent flows[11, 12] Thakare and Parekh [13] employed different turbulence models to explore the energy separation with different working fluids. The results showed that the working fluid with large thermal diffusivity and thermal conductivity caused more significant energy separation effect except for hydrogen. However, the effect of Prandtl number on energy separation was not obvious. Thakare et al. [14] studied the influence of cold mass fraction on energy separation efficiency and stated that when the cold mass fraction is between 0.3 to 0.35, the temperature of the cold air reached the minimum; when the cold mass fraction is 0.68, vortex tube gets the maximum cooling capacity. The numerical simulation by Kandil and Abdelghany [5] found that with the increase of cold mass fraction, the secondary flow would disappear in the tube, which was consistent with the view of Behera et al. [3]. The above researchers studied the vortex tube energy separation using two-dimensional models. More recently, Pourmahmoud et al. [15], Shamsoddini and Nezhad [16] performed numerical investigations on the three-dimensional models of a vortex tube in order to investigate the effects of the number of nozzles on the flow and cooling power of the vortex tube. Muhammad et al. [17] conducted a numerical study based on the experiment which tested the performance of vortex tube controlled by values with different shapes. CFD results were verified by the experimental work, and the maximum absolute errors of the numerical and measured values in two and three dimensions model were 4.12% and 2.3% respectively. Khait et al. [18] carried out entropy generation analysis to determine the irreversible energy conversion in the vortex tube. The standard $k – \varepsilon$ and SAS-SST turbulence models were compared and analyzed from the perspective of thermodynamics. Furthermore, optimization of vortex tube was suggested. Guo and Zhang [19] used the RSM model to study the influence of different cold mass fraction conditions on the flow structure; the energy separation performance was explained based on the vortex breakdown theory. In
numerical solutions, the accuracies of different models for the same flow phenomenon will directly affect the results and prediction. In this article, a complete three-dimensional simulation is accomplished by utilizing different turbulence models, including Spalart-Allmaras, standard $k-\varepsilon$ model, realizable $k-\varepsilon$ model, as well as the SST $k-\omega$ model [20], which is used in cyclone simulation, toward assessing a suitable model for the investigation of energy separation in vortex tube.

2. Mathematical models

2.1. Geometry of models

The geometric model of vortex tube in this study is as close as possible to the experimental model reported by Gao et al. [7]. Figure 3 depicts the structure of the vortex tube, which consists of the following components: a) a square input nozzle (2.7 mm × 2.7 mm, 20 mm length), b) a vortex chamber (30 mm diameter, 11 mm thickness), c) the hot end tube (16 mm diameter, 205 mm length, i.e. $L/D = 12.8$), and d) cold tube (4 mm diameter, i.e. $d_c/D = 0.25$). There are four observation surfaces in the vortex tube, as given in Figure 3.

2.2. Governing equations

Flow in vortex tube is assumed to be compressible and turbulent in steady state. In addition, the hypothesis has made as follows: a) Working fluid is the ideal gas. b) The physical properties of the working air are constant. The simulation of the compressible flows is governed by Navier-Stokes equations include conservations of mass, momentum and energy. In the Cartesian coordinate system, the three-dimensional N-S equation of the tensor form is as follows:

Continuity equation:

$$\frac{\partial \rho}{\partial \tau} + \frac{\partial}{\partial x_j}(\rho u_j) = 0$$  \hspace{1cm} (1)

Momentum equation:

$$\frac{\partial (\rho u_i)}{\partial \tau} + \frac{\partial (\rho u_i u_j)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \delta_{ij} \frac{\partial u_k}{\partial x_k} \right) - \rho \overline{u_i u_j} \right]$$  \hspace{1cm} (2)

Energy equation:

$$\frac{\partial (\rho T)}{\partial \tau} + \frac{\partial (\rho u_i T)}{\partial x_j} = \frac{\partial}{\partial x_j} \left( \mu \frac{\partial T}{\partial x_j} - \rho \overline{u_i T} \right) + S_T$$  \hspace{1cm} (3)

Figure 3. Side view of vortex tube. A-A: cross section at the entrance of the nozzle; B-B: the distance from the cross section of the nozzle is 23.5 mm; C-C: the distance from the cross section of the nozzle is 47.5 mm; D-D: center radial cross section of vortex tube.
Equation of state for ideal gas:

\[ p = \rho RT \] (4)

All of the turbulent transport equations are derived from the Navier-Stokes equations. Transport equation for Spalart-Allmaras model is:

\[
\rho \frac{D\tilde{\nu}}{Dt} = G_{\nu} + \frac{1}{\sigma_{\nu}} \left\{ \frac{\partial}{\partial x_i} \left( (\mu + \rho \tilde{\nu}) \frac{\partial \tilde{\nu}}{\partial x_j} \right) + C_{b2} \rho \left( \frac{\partial \tilde{\nu}}{\partial x_j} \right) \right\} - Y_{\nu} \] (5)

where \( G_{\nu} \) is the production of turbulent viscosity and \( Y_{\nu} \) is the destruction of turbulent viscosity that occurs in the near-wall region due to the wall blocking and viscous damping; \( \tilde{\nu} \) is the molecular kinematic viscosity; \( \sigma_{\nu} \) and \( C_{b2} \) are constants.

Transport equations for the standard k-\( \varepsilon \) model are:

\[
\rho \frac{dk}{dt} = \frac{\partial}{\partial x_j} \left( (\mu + \frac{\mu_i}{\sigma_k}) \frac{\partial k}{\partial x_j} \right) + G_k + G_b - \rho \varepsilon - Y_M \] (6)

\[
\rho \frac{de}{dt} = \frac{\partial}{\partial x_j} \left( (\mu + \frac{\mu_i}{\sigma_e}) \frac{\partial \varepsilon}{\partial x_j} \right) + C_{1e} \frac{\partial \varepsilon}{\partial x_j} + C_{2e} \frac{\varepsilon^2}{k + \sqrt{\nu_e}} + C_{3e} \frac{C_{1e} \varepsilon}{k} (G_k + C_{3e} G_b) \] (7)

where \( G_k \) is the turbulent kinetic energy caused by the average flow velocity gradient, \( G_b \) is the turbulent kinetic energy caused by buoyancy, and \( Y_M \) is the effect of compressible turbulence pulsation expansion on the total dissipation rate. Viscosities of the turbulence coefficient are \( \nu_i = \rho C_{\mu i} \frac{\varepsilon}{k} \); \( C_{1Z} = 1.44 \), \( C_{1Z} = 1.44 \), and \( C_{1Z} = 0.09 \).

Transport equation for SST \( k - \omega \) model is:

\[
\frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial x_i} (\rho k u_i) = \frac{\partial}{\partial x_j} \left[ \Gamma_k \frac{\partial k}{\partial x_j} \right] + G_k - Y_k + S_k \] (8)

\[
\frac{\partial}{\partial t} (\rho \omega) + \frac{\partial}{\partial x_i} (\rho \omega u_i) = \frac{\partial}{\partial x_j} \left[ \Gamma_\omega \frac{\partial \omega}{\partial x_j} \right] + G_\omega - Y_\omega + S_\omega \] (9)

where \( G_\omega \) represents the generation of \( \omega \), \( \Gamma_k \) and \( \Gamma_\omega \) represent the effective diffusivity of \( k \) and \( \omega \), respectively. \( Y_k \) and \( Y_\omega \) represent the dissipation of \( k \) and \( \omega \) due to turbulence. \( S_k \) and \( S_\omega \) are user-defined source terms.

Transport equations for realizable \( k - \varepsilon \) model are:

\[
\rho \frac{Dk}{Dt} = \frac{\partial}{\partial x_j} \left( (\mu + \frac{\mu_k}{\sigma_k}) \frac{\partial k}{\partial x_j} \right) + G_k + G_b - \rho \varepsilon - Y_M \] (10)

\[
\rho \frac{De}{Dt} = \frac{\partial}{\partial x_j} \left( (\mu + \frac{\mu_k}{\sigma_k}) \frac{\partial \varepsilon}{\partial x_j} \right) + \rho C_1 S\varepsilon - \rho C_2 \frac{\varepsilon^2}{k + \sqrt{\nu_e}} + \frac{\nu}{k} C_{1e} \frac{\varepsilon}{k} (G_k + C_{3e} G_b) \] (11)

where \( C_1 = \max \left[ 0.43, \frac{\nu}{\nu + 3}\right], \sigma_k = 1.0 \), and \( \sigma_\varepsilon = 1.2 \).

3. Boundary conditions and solver

The boundary conditions for this problem include: a) the type of inlet is pressure-inlet that the total pressure makes the static pressure reach 0.65 MPa (gauge pressure); the working fluid is nitrogen and the inlet total temperature reaches 285.6 K; b) the type of cold-exit is pressure-outlet, and the magnitude of the total pressure is kept with standard atmospheric pressure; c) the type of hot-exit is also pressure-outlet, and the total pressure is adjusted to keep the cold mass fraction around 0.27 in different turbulent models. PREssure STaggering Option (PRESTO)
Scheme is used for discretization of pressure, which is reported to perform better when the steep pressure gradient is involved in the swirling flows.

4. Results and discussions

4.1. Pressure distribution

When the gas injects into the vortex chamber through the nozzle at high velocity, the high-pressure gas expands rapidly causing a sudden drop in pressure. The temperature drop near the injection induced by adiabatic expansion can be expressed as:

$$P_{c_1}^{1-\gamma} = P_{c_2}^{1-\gamma} \frac{T_1^{1-\gamma}}{T_2^{1-\gamma}}$$

where $\gamma$ is the specific heat capacity ratio, which is generally 1.4 for diatomic ideal gas.

The distribution of the static pressure field in the vortex tube obtained by different turbulence models is shown in Figures 4 and 5. As can be found in Figure 4, all models predict similar distribution of static pressure in the peripheral region, whereas there are evident differences in the center region. The blue region in Figure 4(d) is larger than others, which means the low-pressure area calculated by realizable $k-\varepsilon$ is broader than others. On the other hand, the gas pressure increases with increasing $r/R$. The results support the theory of temperature drop near the injection caused by adiabatic expansion. It is also seen from Figure 5 that the pressure in the tube is lower than the inlet pressure, which indicates that the flow in the tube is accompanied by an expansion process.

4.2. Velocity distribution

4.2.1. Tangential velocity

Compared with the axial and radial velocities, the tangential velocity is usually the largest in value, which can reflect the flow direction of particles in the vortex tube. The fast-tangential velocity also leads to a strong centrifugal force. The distribution of the tangential velocity in the vortex tube predicted by different turbulence models is provided in Figure 6. It was found that the tangential velocity of gas near the nozzle can be close to the speed of sound. In the same axial section, the maximum tangential velocity appears near the wall and the velocity decreases gradually toward the center of the tube. As the distance from the inlet increases, the tangential velocity of the gas decreases. It is probable that the kinetic energy is gradually converted into thermal energy by viscous dissipation and the friction between the gas and the wall during the
vortex movement. Furthermore, a proper increase of $L/D$ value is beneficial to improve the performance of energy separation. The tangential velocity distributions obtained by different turbulence models showed differences at the near hot end. The gas swirl velocity in the hot gas region obtained by Spalart-Allmaras model was significantly lower than that of other models. The hot gas region predicted by realizable $k-\varepsilon$ model maintains a high swirl velocity.

The tangential velocity distribution of C-C cross section was compared with experimental data of Gao et al. [7], Takahama [21] and Bruun [22]. All working conditions have been listed in Table 1. All data should be nondimensionalized and the non-dimension tangential velocity is defined as:

$$\hat{v}_t = \frac{v_t}{\text{max}(v_t)}$$

where $V_t$ is the tangential velocity, and $\text{max}(V_t)$ means the maximum tangential velocity at the same section.

Figure 7 illustrates the difference between the simulation and experimental results, especially close to the center of the tube. There are many reasons for this difference. Firstly, the influence of the Pitot tube probe on the flow field in the experiments cannot be ignored. Secondly, the direction of measurement affects the velocity results. Finally, the difference in the structure and the measurement interfaces of the vortex tube bring interference to the comparison. Although there is a difference between the results, the tangential velocity in the one interface is consistent with
the distribution trend. When $r/R$ is in the range of $0 \sim 0.7$, the gas inside the tube remains approximately the same as the angular velocity. The tangential velocity of the gas flows reaches the highest point at $r/R = 0.8 \sim 0.95$. As the $r/R$ increases further, the gas velocity drops. The simulation results of Pourmahmoud et al. [15] and Thakare [23] had similar conclusions. Fulton [24] suggested that the angular velocity at the peripheral location was lower than that at the central location and an almost free vortex was formed near the inlet nozzle. Under the action of internal friction, the fluid in a cross section will rotate at the same angular velocity, which is similar to the rotational motion of a rigid solid. The diffusion energy transmits from the outer layer to the inner fluid, while the mechanical energy is transferred conversely. The outer gas gains more kinetic energy than the loss of internal energy and the inner gas loses kinetic energy, which leads to the temperature separation in the vortex tube. The model of this mechanism is represented in Figure 8.

Figure 6. Tangential velocity distribution of D-D cross section with different turbulence models (m/s).

Table 1. Working condition used by different experiments (length unit: mm).

<table>
<thead>
<tr>
<th>Author</th>
<th>D</th>
<th>L</th>
<th>Number of nozzles</th>
<th>dc</th>
<th>mc/min</th>
<th>Z/D</th>
</tr>
</thead>
<tbody>
<tr>
<td>Gao et al.</td>
<td>16</td>
<td>205</td>
<td>1</td>
<td>4</td>
<td>0.27</td>
<td>2.97</td>
</tr>
<tr>
<td>Takahama</td>
<td>52.8</td>
<td>7920</td>
<td>4</td>
<td>–</td>
<td>0.5</td>
<td>0.5</td>
</tr>
<tr>
<td>Bruun</td>
<td>94</td>
<td>520</td>
<td>4</td>
<td>35</td>
<td>0.23</td>
<td>1.27</td>
</tr>
</tbody>
</table>
4.2.2. Axial velocity

Figure 9 displays the axial velocity distribution predicted by different turbulence models. The blue area represents the gas flow to the hot end, the red and green areas represent the gas flow to the cold end. The simulation results indicate that there is a curved surface between the center and wall with zero axial velocity. The vortex tube is divided by the curved surface into two parts. The axial velocity near the center is positive and flows to the cold orifice, which is known as internal cooling vortex. The other part of the axial velocity is negative, and the flow direction is to the hot end, which is called external heat vortex. However, the result of the Spalart-Allmaras model is an exception: after the air is injected into the tube, the curved surface gradually gets closer to the center, and eventually into a point. There is a secondary circulation at the end of the hot end.

The dimensionless axial velocity is defined as:

$$\tilde{v}_a = \frac{v_a}{\max(v_t)}$$ (13)

where \(v_a\) is the axial velocity, m/s.

As shown in Figure 10, the axial velocity in the vortex tube usually does not exceed 0.4 of the maximum tangential velocity in any cross section. The maximum axial velocity of the inner vortex occurred near the center, which could be caused by the pressure gradient. The maximum axial velocity of the outer vortex is near the wall because of the energy transfer. In addition, it should be added that the simulation results of SST \(k-\omega\) model and realizable \(k-\varepsilon\) model are closer to the experimental data than the other two models.
4.2.3. Radial velocity

Most of the radial velocity values are distributed in a range from $-20 \text{ m/s}$ to $+20 \text{ m/s}$, which is less than the axial velocity and tangential velocity. Although the radial velocity can be considered as negligible compared with the other velocities, the radial velocity can still reveal some flow

Figure 9. Axial velocity distribution of D-D cross section with different turbulence models (m/s).

Figure 10. Comparison of the dimensionless tangential velocity.

4.2.3. Radial velocity

Most of the radial velocity values are distributed in a range from $-20 \text{ m/s}$ to $+20 \text{ m/s}$, which is less than the axial velocity and tangential velocity. Although the radial velocity can be considered as negligible compared with the other velocities, the radial velocity can still reveal some flow
characteristics. According to Figure 11, the maximum radial velocity is concentrated in the tube center and the radial velocity is close to zero in the periphery zone. At the center, the average radial velocity is not zero, and the distribution of radial velocity is nearly axisymmetric, which indicates that the vortex flow pattern is likely to be a wave oscillation pattern.

4.3. Temperature field

The total temperature distribution of D-D cross section is indicated in Figure 12. It shows that all turbulence models can reflect the energy separation phenomena. The gas expands after being injected into the vortex chamber, and the static temperature of the gas decreases rapidly. Since the velocity is close to the speed of sound, the total temperature in the vortex tube is still higher. In the process of gas moving to hot end in the form of a vortex flow, the internal energy is dissipated as heat as a result of the friction between the wall and gas flow, which leads to the total temperature reduction in this region. When the gas flows from the hot end to the cold end, the internal energy will be reduced again due to the kinetic energy transfer. The gas becomes colder in the center of the vortex tube.

The outlet temperature of various turbulence models is listed in Table 2. It can be found from Table 2 that the best energy separation effect can be obtained under the SST $k - \omega$ model, however, the results seriously deviate from the experimental results. On the contrary, the realizable $k - \varepsilon$ model simulation is quite close to the experimental data.

![Figure 11. Radial velocity distribution of D-D cross section with different turbulence models (m/s).](image)
4.4. Streamline field

Figure 13 shows the streamlines in the D-D plane obtained from different models. The streamlines were calculated from the tangential velocity and axial velocity. It can be seen that there exists a large number of secondary circulations.

The results based on the Spalart-Allmaras turbulence model describes that the secondary circulations are mainly concentrated in the first half of the tube (close to the cold exit). After entering the tube, the flow direction of gas changes to the cold end near the secondary circulations. The positions of the secondary circulations calculated by other models are similar. The cold end gas forced back in the middle of the vortex tube based on the standard $k-\varepsilon$ model. Both SST $k-\omega$ model and realizable $k-\varepsilon$ model indicate the reversed flow occurs at the hot end.

5. Conclusion

A numerical study has been carried out to investigate the energy separation mechanism and flow phenomena within a vortex tube using four different turbulence models: Spalart-Allmaras model,
standard $k-\varepsilon$ model, SST $k-\omega$ model, and realizable $k-\varepsilon$ model. The simulation results have been compared with the available experimental data. It was demonstrated that all turbulence models can simulate temperature separation. The simulation results of different turbulence models have similar pressure field and velocity field profile. The sudden gas expansion in the vortex chamber causes the pressure drop, which is the dominated reason for the low temperature of the cold end. The pressure in other parts of the tube is lower than the inlet pressure, and the adiabatic compression is not the cause of the high temperature at the hot end. Friction between gas and wall surface and the energy transfer between different laminar flows are the main factors of the high temperature of peripheral gas and the low temperature of the center zone. There are significant differences in the simulation results with the various turbulence models. Based on the analysis mentioned, the realizable $k-\varepsilon$ model is better than other models in terms of the structure and working conditions of the vortex tube in this study.

**References**


