International Journal of Modern Trends in Engineering and Research
www.ijmter.com
e-ISSN No.:2349-9745, Date: 2-4 July, 2015

3D CFD Analysis of Vortex Tube

Hitesh R. Thakare [1], Aniket Monde [2], A. D. Parekh [3]

[1]Department of Mechanical Engineering, S. V. N. I. T., Surat, Gujarat, India, thakare.hitesh@gmail.com
[2] Department of Mechanical Engineering, S. V. N. I. T., Surat, Gujarat, India, aniketmonde22@gmail.com
[3] Department of Mechanical Engineering, S. V. N. I. T., Surat, Gujarat, India, adp@med.svnit.ac.in

Abstract—This paper is related to CFD study carried out on counter—flow vortex tube. The CFD model is a three-dimensional (3D) model that utilizes standard k-epsilon turbulence model. This CFD study has been used to understand the fluid behaviour inside the vortex tube. The objective of this work is the demonstration of the successful use of CFD in this regard, thereby providing a powerful tool that can be used to optimize vortex tube design as well as assess its utility in the context of new applications. Results of present CFD study are in good agreement with experimental results.
Keywords- Counter flow vortex tube, Turbulence model, Swirl velocity, Temperature separation

I. INTRODUCTION

A vortex tube, invented by Ranque [1], is a simple and essentially compact device having capability to divide input stream of pressurized air into simultaneous streams of hot and cold air at reduced pressure. When pressurized air is admitted into the vortex tube in tangential direction, it attains high swirl velocity. This motion sets up a vortex type of flow inside the tube. Due to internal mechanism, one of the streams exiting the tube is at lesser, while other stream is at higher temperature than that of input air stream. This phenomenon is known as energy separation or temperature separation. By controlling the position of control valve located on the hot side of the tube, mass of cold and hot air coming out of vortex tube can be controlled. The magnitude of temperature separation is affected with change in mass of cold gas extracted i.e. cold mass fraction, the pressure of input stream and geometry of vortex tube.

The distinctive feature of vortex tube is that such temperature separation effect gets produced almost instantaneously, even without any moving parts or any chemical reaction inside the tube. Absence of moving parts enables vortex tube to have very low maintenance, thereby extended service life. Due to ability to produce instant cold and hot air, vortex tube finds use in important applications such as comfort suits of mine workers and spray painters, spot cooling of electronic components in limited space, gas liquefaction 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. For 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 this point of ambiguity, CFD can significantly help the researchers. Shamsoddini et al. [3] observed that cooling power increased with increase in nozzle number.

Cockerill [4] 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 study of Dutta et al. [5] reported 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. CFD study of Baghdad et al. [6] concluded that temperature separation prediction by RSM model was better while other models over predicted the temperature separation effect.

This paper is aimed to report the characteristic of flow physics parameters inside vortex tube. Results of present CFD study are compared with experimental and CFD results of Skye et al [7].

II. MATHEMATICAL MODELLING

The governing equations solved by Fluent™ 6.3.26 are equation of continuity, momentum, energy and equation of state, given below:-

$$\frac{\partial \rho}{\partial t} + \nabla.\ \rho\vec{v} = S_m \tag{1}$$

$$\frac{\partial}{\partial t}\ \rho\vec{v} + \nabla.\ \rho\vec{v}\vec{v} = -\nabla p + \nabla.\ \bar{\bar{\tau}} + \rho\vec{g} + \vec{F}, \tag{2}$$

$$\frac{\partial \rho h_o}{\partial t} - \frac{\partial P}{\partial t} + \mathrm{div}(\rho h_o \mathrm{U}) = \mathrm{div}\ \lambda\, \mathrm{grad}\, T_s \tag{3}$$

$$P = \rho R T_s \tag{4}$$

Transport equations in Standard k-ε model are for turbulence kinetic energy, *k*, and its rate of dissipation, ε, given below:

$$\frac{\partial}{\partial t}\ \rho k + \frac{\partial}{\partial x_i}\ \rho k u_i = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + G_k + G_b - \rho\varepsilon - Y_M + S_k \tag{5}$$

$$\frac{\partial}{\partial t}\ \rho\varepsilon + \frac{\partial}{\partial x_i}\ \rho\varepsilon u_i = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + C_{1\varepsilon}\frac{\varepsilon}{k}\ G_k + C_{3\varepsilon}G_b\ - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} + S_\varepsilon \tag{6}$$

III. BOUNDARY CONDITIONS

For present CFD study, geometry of vortex tube as well as boundary conditions of experimental study of Skye et al. [7] has been used. Details about this have been avoided here for the sake of brevity and can be found in the work of Skye et al. [7]. The structured grid has been created using Gambit and 2.4.6 while mathematical equations have been solved using commercially available code Fluent™ 6.3.26. The meshed vortex tube is depicted in Fig. 1.

All the CFD simulations have been performed in steady state mode using pressure based implicit solver. The standard k–ε model has been used to model the turbulent flow inside the tube. The flow inside the vortex tube is compressible. The inlet of vortex tube is considered as mass flow inlet. The hot end and cold end are considered as pressure outlet. The cold end is open to atmosphere. Gravity is neglected. Pressure on hot end is varied to obtain various cold mass fractions. SIMPLE algorithm is used for Pressure-Velocity Coupling. Default values of Under Relaxation Factors have been used. To set the convergence criteria, default values of residuals of the order of 10^{-6} were used for energy while it was 10^{-3} for all other quantities. Wall of the vortex tube is assumed to be stationary and adiabatic with no-slip boundary condition.

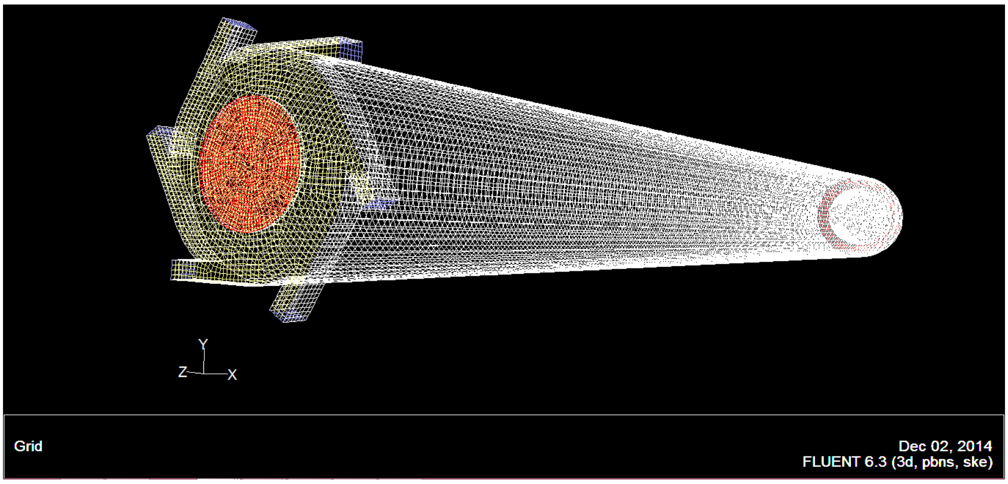

Fig 1: Structured Mesh of Vortex Tube Used In Present Simulation

IV. GRID INDEPENDENCE STUDY

Grid independence of solution has been checked using structured grid of different sizes i.e. with 32215, 98510 and 152553 cells. This check is necessary so as to eliminate the errors arising due to coarseness of the grid. This is shown in Fig. 2.

As observed from Fig. 2, for all the grid sizes, magnitude of temperature separation is nearly same at all values of cold mass fractions. Hence, further study has been carried out for grid size of 152553 cells.

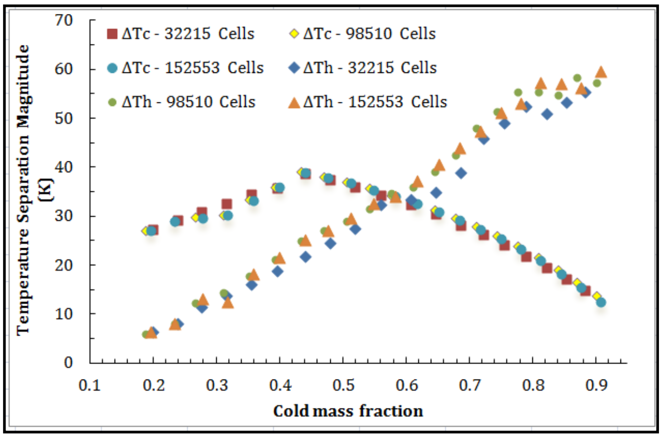

Fig 2: Effect of Grid Size on Temperature Separation Magnitude

V. VALIDATION

Results of present CFD study are compared with experimental and CFD results of Skye et al. [4] for validation and authentication. As observed from Fig. 3, results of present CFD study are in good agreement with experimental results. It implies that simulation methodology followed in present work is appropriate.

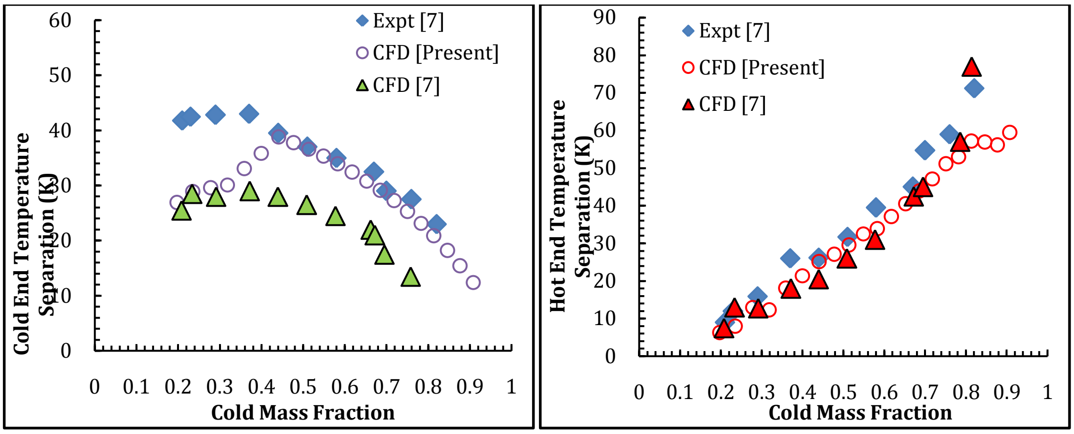

Fig 3: Comparison of CFD results (a) Cold End (b) Hot End Temperature Separation

VI. RESULTS AND DISCUSSION

This section presents nature of flow parameters inside vortex tube. Fig.4 illustrates contours of swirl velocity inside vortex tube. Swirl velocity is maximum at nozzle inlet. It is found to decrease with an increase in axial distance i.e. as flow of air progresses towards hot end. Swirl velocity is minimum near the hot end. Along the radial direction of the tube, the swirl velocity is observed to have the minimum magnitude near the tube axis. Throughout the tube, inner core is observed to have negligible swirl velocity.

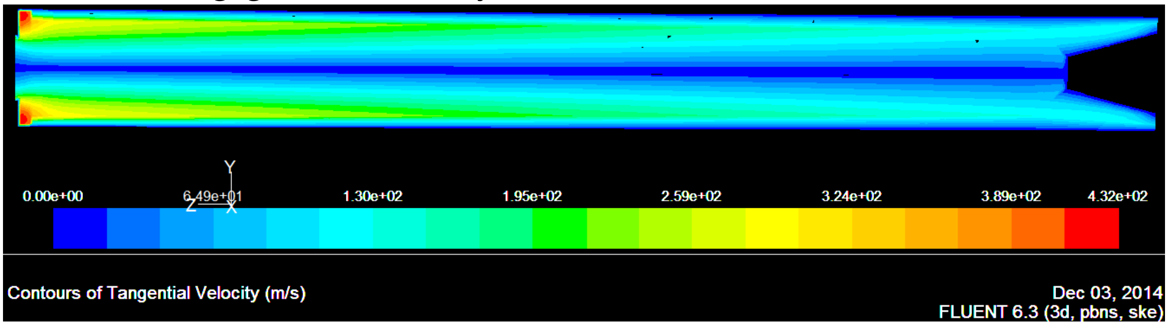

Fig 4: Contours of Tangential or Swirl Velocity inside Vortex Tube

Fig. 5 shows variation of swirl velocity at different axial locations. At all axial locations swirl velocity increases with radius i.e. towards the periphery. Near the wall of the tube, swirl velocity suddenly drops down to zero due to no-slip boundary condition at the wall. Magnitude of swirl velocity decreases with increase in axial distance from cold end.

Fig. 6 shows variation of axial velocity inside vortex tube. It is observed that the maximum axial velocity in hot region is near the tube wall and the direction of the flow near the wall is towards the hot end exit. The direction of the flow along the axis is towards the cold end exit. Also, the maximum value of the axial velocity in hot region decreased with increasing axial distance. The axial velocity in the cold core was found to increase with a decrease in the axial distance i.e. as we move from hot exit to cold exit. The drag force caused by the difference of pressure between flow field and cold end exit will continuously act on particles moving towards the hot end. When the particle is not left with any momentum to flow against this pressure gradient, its axial velocity ceases to zero and later on reverse its direction of flow, by moving towards the cold end exit. Further acted by the differential pressure, the particle expands causing to considerable increase in axial velocity in reverse direction.

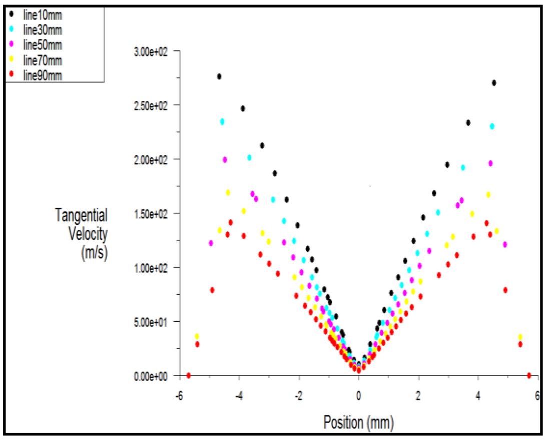

Fig 5: Swirl Velocity at Different Axial Locations

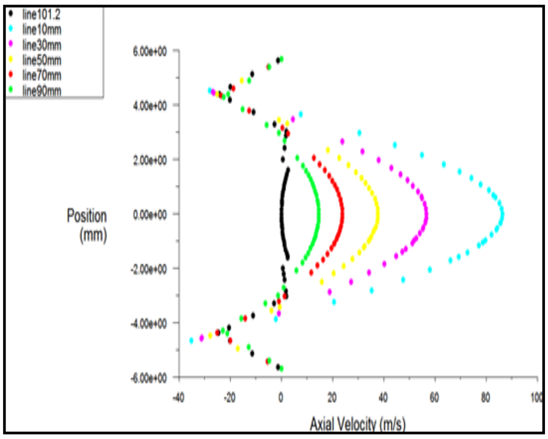

Fig 6: Variation of Axial Velocity Inside Vortex Tube

Fig. 7 shows streamline structure inside vortex tube. First, the air introduced in the vortex tube via the inlets positioned at a given angle of the main pipe, creates vortices in the internal flow. The model selected is capable of predicting this flow behaviour which is expected as this is imposed mainly by the angular position of the inlet sections. In addition, the figure illustrates the existence of two vortices, both; rotating in the same direction with the same angular velocity. This is consistent with observation of Baghdad et al [6], shown in Fig. 8. The first is a free vortex that characterizes the outer regions and tends to escape through the hot exit, whereas the second one is a forced vortex representing the reverse flow that appears in the core region and moves back towards the cold exit crossing all the hot tube and the vortex chamber centre. It can also be noticed that the reverse flow starts moving towards the cold exit at a specific location along the main tube.

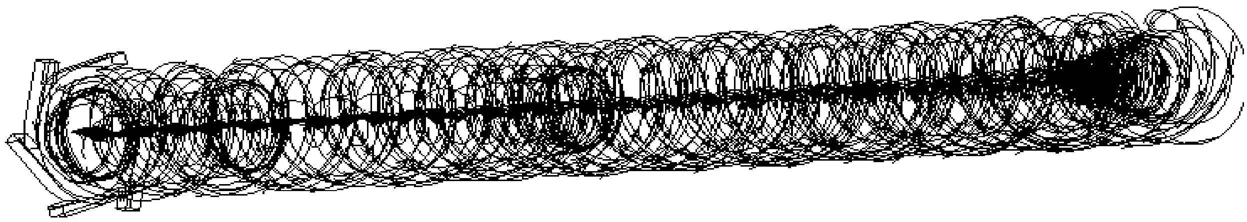

Fig 7: Streamline Structure inside Vortex Tube [Present]

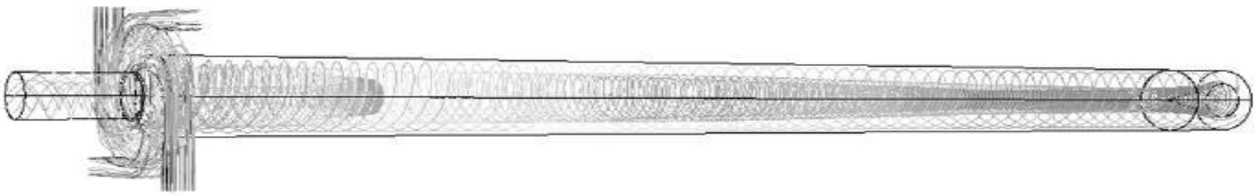

Fig 8: Streamline Structure inside Vortex Tube [6]

Fig. 9 illustrates the static temperature as a function of radius at different axial locations and shows that the static temperature decreases in the radial direction (as r gets larger), except very near the tube inlet. The decrease in static temperature in radial direction creates the temperature gradient for the heat energy to flow radially. This temperature gradient is maintained throughout the vortex tube length by the velocity gradient in radial direction. Due to no slip boundary condition at the wall, static temperature increases very near to wall because of conversion of kinetic energy into thermal energy. In this region heat transfer will be in negative radial direction i.e. radially inwards.

Fig. 10 shows magnitude of total temperature inside vortex tube at different axial locations. Total temperature increases with increase in axial distance. The total temperature profiles depend upon distribution of kinetic energy in vortex tube. Comparing the total temperature and the swirl velocity profiles, we observe that the low temperature zone in the core coincides with the negligible swirl velocity zone. Fig. 11 shows contours of total temperature inside vortex tube observed during our study. The phenomenon of temperature separation is clearly visible in this contour plot. The peripheral flow is warm and core axial flow is cold relative to the inlet flow. Maximum total temperature is observed near hot end of tube. Magnitude of temperature separation decreases with increase in axial distance.

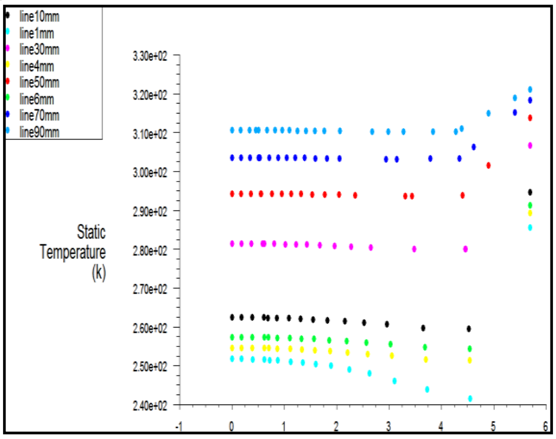

Fig 9: Static Temperature inside Vortex Tube

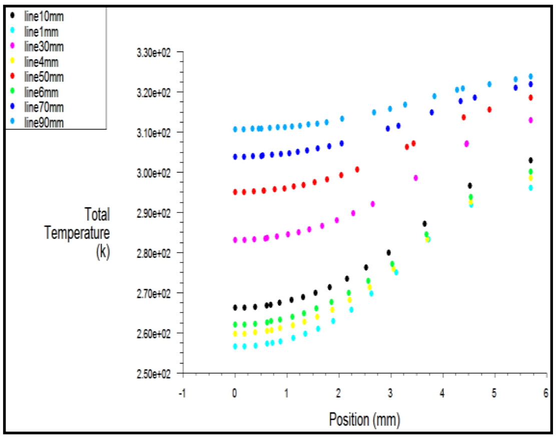

Fig 10: Total Temperature Inside Vortex Tube

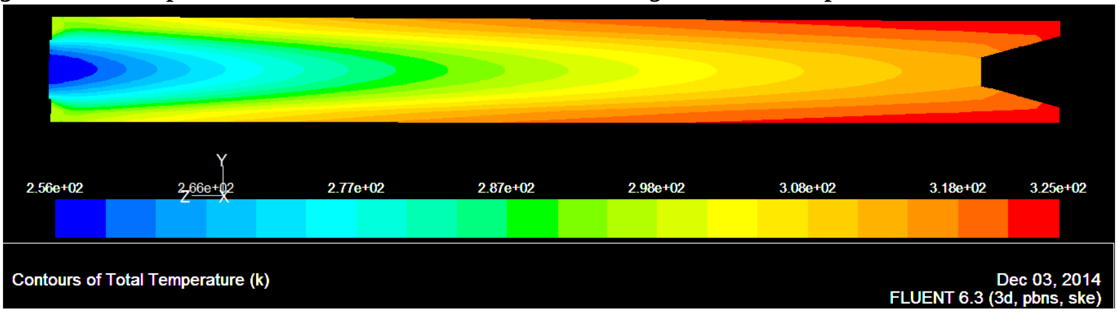

Fig 11: Contours of Total Temperature inside Vortex Tube

CONCLUSION

CFD simulation of vortex tube was attempted in order to understand nature of flow physics inside the tube. It is concluded that CFD results are in good agreement with experimental results. Swirl velocity is the largest component of flow velocity. Profile of axial velocity clearly indicates presence of flow reversal inside the tube. Core axial flow is colder while peripheral flow is hotter than inlet air. Static temperature decreases towards periphery which can cause heat transfer from core axial to peripheral air mass.

REFERENCES

[1] G. L. Ranque,Method and apparatus for obtaining from a fluid under pressure two currents of fluids at different temperatures, U.S. Patent No. 1952281, 1934.

[2] B. Ahlborn, S. Groves, Secondary Flow in Vortex Tube, Fluid Dynamic Research, 21 (1997) 73–86.

[3] R. Shamsoddini, A. H. Nezhad, "Numerical analysis of the effects of nozzles number on the flow and power of cooling of a vortex tube", Int. J. Refrig. vol. 33 pp. 774–782, 2010.

[4] T. T. Cockerill, "Fluid mechanics and thermodynamics of a Ranque-Hilsch vortex tube, Masters Thesis, University of Cambridge, 1995.

[5] T. Dutta, K.P. Sinhamahapatra, S. S. Bandyopdhyay, "Comparison of different turbulence models in predicting the temperature separation in a Ranque–Hilsch vortex tube, Int. J. Refrig. vol. 33 pp. 783–792, 2010.

[6] M. Baghdad, A. Ouadha, O. Imine, Y. Addad, "Numerical study of energy separation in a vortex tube with different RANS models", Int. J. The. Sci. vol. 50 pp. 2377-2385, 2011.

H. M. Skye, G. F. Nellis, S. A. Klein, "Comparison of CFD analysis to empirical data in commercial vortex tube", Int. J. Refrig. vol. 29 pp. 71–80, 2006.