Experimental and Computational Analysis of Vortex Tube

Bhavesh Parkhe
Student, Don Bosco Institute of Technology, Mumbai

Dr. Prasanna Nambiar
Head of the Department, Mechanical Engineering,
Don Bosco Institute of Technology, Mumbai

Abstract

This project presents a comparison between the performance predicted by a computational fluid dynamics (CFD) model and experimental measurements taken using a vortex tube with specifications similar to the commercial one. Specifically, the measured exit temperatures into and out of the vortex tube are compared with the CFD model. The data and the model are both verified using global mass and energy balances. The CFD model is a three-dimensional (3D) steady state model (with swirl) that utilizes the standard k-epsilon turbulence model. The objective of this study is to observe the temperature separation phenomenon within the vortex tube and validate it with an appropriate experimental model.

Keywords – Vortex Tube, RHVT, CFD, k-epsilon

Introduction

The Ranque-Hilsch Vortex Tube (RHVT) is an ingenious invention credited to both George Joseph Ranque and Rudolf Hilsch, who contrived the device independently during war torn Europe in the 1940's. Intensive experimental and analytical studies of Ranque–Hilsch effect began since then and continue even today. According to these studies when a vortex tube is injected with compressed gas through tangential nozzles into its scroll chamber, a strong circular flow field is established. This vortex in the inlet area causes pressure distribution of the flow in radial direction. As a result a free vortex is produced as the peripheral warm stream and a forced vortex as the inner cold stream. Schematic show in fig 1.

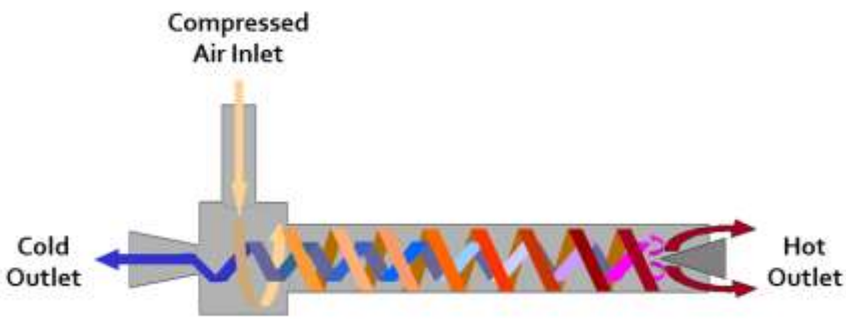

Figure 1 : Working Principle of Vortex Tube

It has been revealed that experimental procedures make it difficult to analyse the parameters owing to a highly turbulent flow. Recent efforts have successfully utilized computational fluid dynamics (CFD) modelling to explain the fundamental principles behind the energy separation produced by the vortex tube.

Therefore, the objective of this paper is to establish a correlation between the experimental and computational analysis. The results will be verified with previous research papers. An experimental model with appropriate dimensions was fabricated for the analysis. The model was similar to that worked on by previous researchers so as to establish a common ground for comparison. A computational geometry of the same dimensions as that of experimental model was made for the analysis.

Experimental Analysis

For previous investigations [1],[2],[3],[5], computational as well as experimental, the Exair 708 slpm model was preferred by many scientists to establish a standard. Thus, to maintain the standard, an experimental model was fabricated similar to the dimensions of ExAir 708 slpm as shown in fig.. A conical obstruction at the hot exit was prepared to suit the computational analysis.

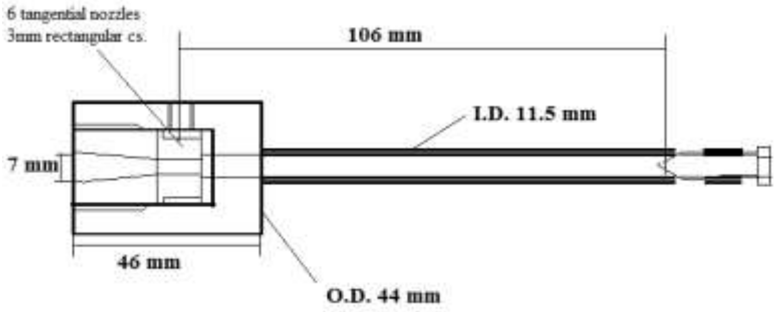

Figure 2 : Dimensions of Vortex Tube

Vortex tube geometry summary

Measurement	Value
Working tube length	10.6 cm
Working tube I.D.	1.14 cm
Nozzle height	0.97 mm
Nozzle width	1.41 mm
Nozzle total inlet area (A_n)	8.2 mm^2
Cold exit diameter	6.2 mm
Cold exit area	30.3 mm^2
Hot exit diameter	11.0 mm

Table 1: Dimensions of Components

The dimensions of the internal components of Exair 708 slpm were measured by Skye [5] in his analysis. The dimensions shown in fig.2 (also mentioned in table 1) will be required for the computational model.

Experimental Setup

To record the measurements of various parameters the following setup was adopted. Measurement of volume flow rate was taken using rotameter at the inlet and cold exit. Gauge pressure was measured using bourdon's tube pressure gauge. Temperatures at inlet and both exits were measured using thermocouples. The schematic diagram of the setup is shown in fig 3. Following are the details of the equipments used

- Pressure Regulator – Ball Type Valve
- Pressure Gauge – Range 0-7 bar (upto 100psi)
- Rotameter – Range 0 to 500 LPM Air.
- Thermocouple Junction – J type

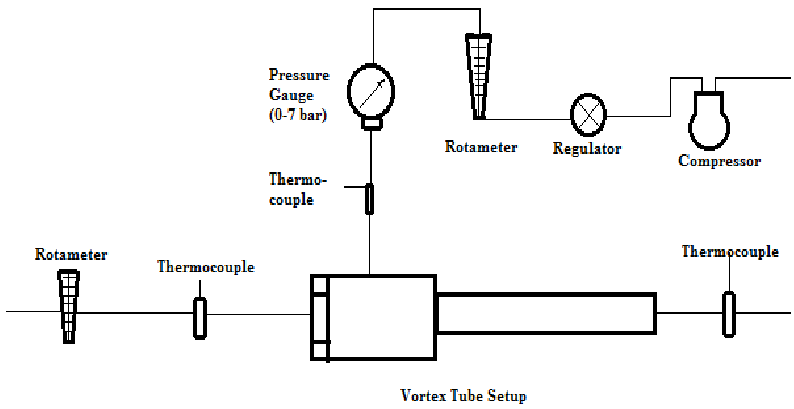

Figure 3: Experimental Setup

For the experiment, the compressor was kept running for 20 minutes to obtain a steady pressure of 10 bar in the tank. The line pressure and then the inlet pressure was controlled using a ball type valve. Minimal inlet pressure of 6 bar was maintained throughout the experiment. Inlet temperature of compressed air is maintained at 26^{o}C.

Computational Analysis

The current advent of technology has made it possible to simulate processes difficult to analyse experimentally. The CFD analysis is done using the ANSYS Fluent package. The modelling is done using ICEM CFD to provide a better mesh and periodic linking. ICEM CFD is preferred for making geometries because of its wide range of options and the ability to mesh complex geometries.

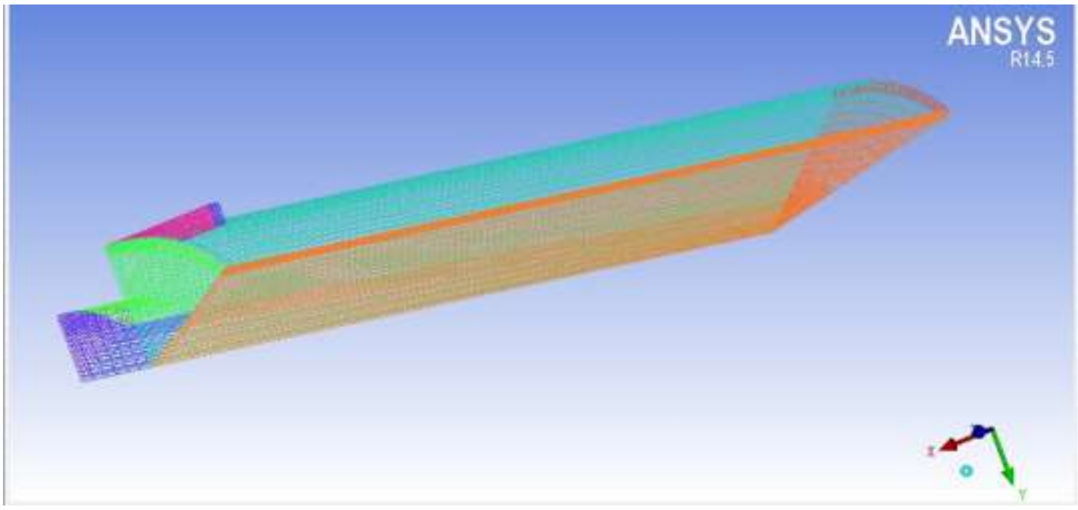

Figure 4: ICEM mesh of computational model

The model is a 3 dimensional (60^o section) as shown in fig.4 similar to analysis by Behera [1]. The cold exit is extended observe the effect of backflow. The mesh is unstructured with finer grids near the inlet and both the exits. Dimensions are similar to the experimental model.

The standard k-epsilon turbulence model is used with steady state axisymmetric modelling. The standard k-epsilon was found to give better result in previous attempts [1], [5], [6].

Boundary Conditions (set depending upon previous attempts and experimental conditions) Inlet was modelled as a mass flow inlet. Total mass flow rate 8.6 g/s and inlet temperature 298 K. Cold Outlet was modelled as a pressure outlet with ambient conditions.

Hot Outlet pressure was set depending on temperature requirements at cold end. Average back pressure of 0.5 bar (gauge) at hot exit.

No-slip boundary condition is enforced on all the walls of vortex tube. Possibility of a backflow at the cold exit was revealed in previous attempts. Hence, the backflow temperature is defined as ambient temperature (300K).

Results & Discussion

Temperature Profiles by computational analysis are as shown in the fig.5. The hot and cold outlets produced air flow as per their respective behaviour. The vortex tube temperature profiles also matched the experimental methods. The hot and cold outlet showed positive exit flow with negligible backflow. The minimum cold exit temperature was upto 252 K and hot exit temperature is upto 323 K.

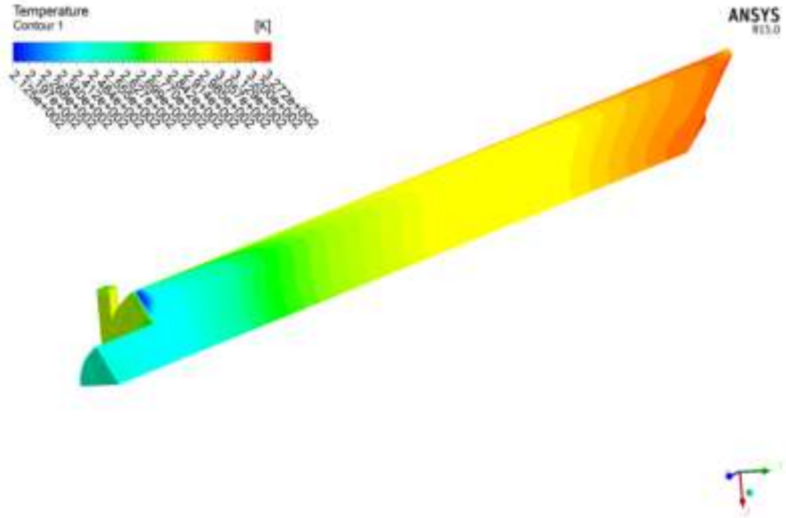

Figure 5: Temperature Contours

The direction of flow conformed with the previous investigations and the assumed internal flow field. The velocity vectors at the cold end are shown in fig. 6.

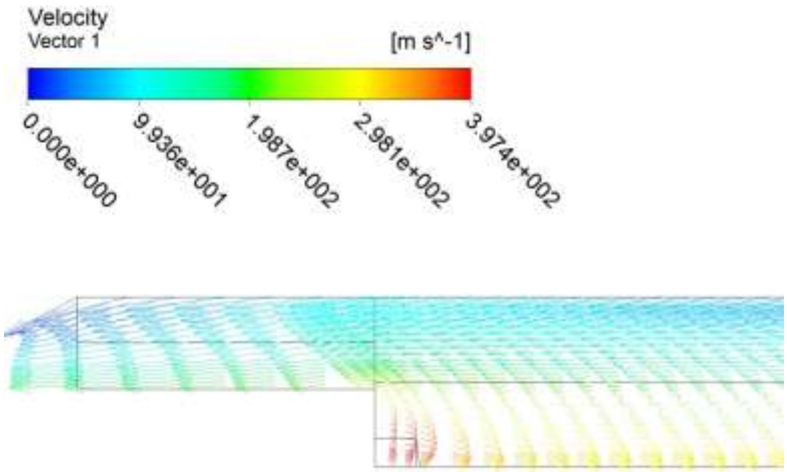

Figure 6 : Velocity Vectors

Minimum temperature at the cold exit for the experimental model was measured to be 253 K. At the flow rate of 48 LPM and annular inner diameter of 11 mm. Fig. 7 shows the comparison between computational and experimental performance of the model. The results given by computational analysis were slightly superior as compared to the experiment. This can be explained due to leakages in the model due to several intrusive methods.

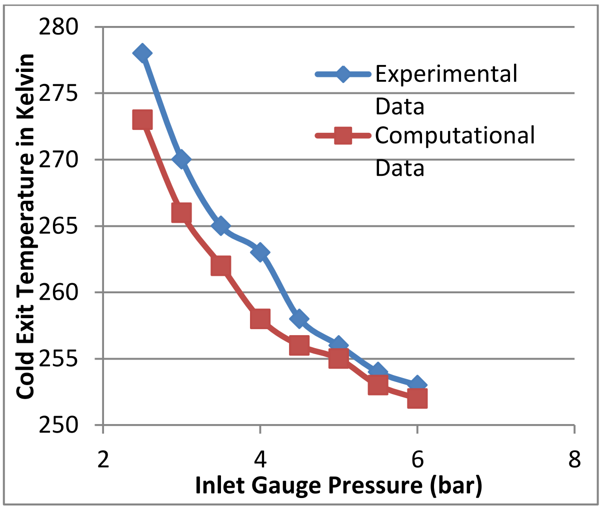

Figure 7: Comparison between Computational and Experimental Data

Owing to the leakages in the model due to various intrusive measurement techniques, the relation between the cold exit flows and the temperatures was found to be unreliable. Therefore the flow rate and the cold fraction parameters have not been included in this paper for comparison. However, the pattern of change in the experimental model is similar to the computational ones in the previous papers. The variation of temperature with respect to the cold flow rate has been presented for viewing the change and must not be relied on for the actual values. The graph can be compared with the results in [1],[4],[5],[6].

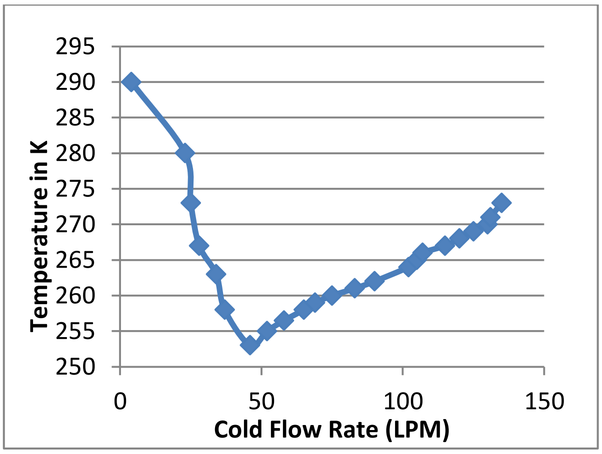

Figure 8: Temperature vs. Cold Flow rate

Conclusions

This experiment was performed with an objective to establish a correlation between computational simulation and actual conditions. The comparison between the CFD model and the measured experimental data yielded promising results relative to the model's ability to predict the power separation. Following were the conclusions with respect to the results.

1. The comparison between the CFD model and the measured experimental data yielded promising results relative to the model's ability to predict the temperature separation.
2. CFD can be used for analysis of a vortex tube. The k-epsilon turbulence model has been successfully tested for the vortex tube.
3. Increasing the cold flow initially reduces the temperature at cold end. After reaching an optimum temperature it starts increasing again.

References

1. U. Behera, P.J. Paul, S. Kasthurirengan, R. Karunanithi, S.N. Ram, K. Dinesh, S. Jacob, CFD analysis and experimental investigations towards optimizing the parameters of Ranque–Hilsch vortex tube
2. N. Pourmahmoud, A. Hassan Zadeh, O. Moutaby, A. Bramo, Computational Fluid Dynamics Analysis Of Helical Nozzles Effects On The Energy Separation In A Vortex Tube, Thermal Science, Year 2012, Vol. 16, No. 1, pp. 151-166
3. A. Bramo And N. Pourmahmoud, Computational Fluid Dynamics Simulation Of Length To Diameter Ratio Effects On The Energy Separation In A Vortex Tube, Thermal Science, Year 2011, Vol. 15, No. 3, pp. 833-848
4. Anayet U. Patwari, N.M Reehan, Adib Bin Rashid and Ashikur Rahman, Computational Analysis Of The Effect Of Vortex Parameters In A Vortex Tube
5. H.M. Skye, G.F. Nellis, S.A. Klein, Comparison of CFD analysis to empirical data in a commercial vortex tube, International Journal of Refrigeration 29 (2006) 71–80
6. Ronán Oliver, Fergal Boyle, Anthony Reynolds, Computer Aided Study of the Ranque–Hilsch Vortex Tube Using Advanced Three-Dimensional Computational Fluid Dynamics Software, Proceedings of the 6th WSEAS International Conference on Applied Computer Science, Tenerife, Canary Islands, Spain, December 16-18, 2006, Page 478