

Advanced Research on Computational Analysis of Optimum Vortex Tube and Experimental Validation

Piyush Kute¹, Rohan Kulkarni², Nilesh Patil³U.G. Student, Department of Mechanical Engineering, NBN School of Engineering, Pune, Maharashtra, India¹U.G. Student, Department of Mechanical Engineering, NBN School of Engineering, Pune, Maharashtra, India²Assistant Professor, Department of Mechanical Engineering, NBN School of Engineering, Pune, Maharashtra, India³

ABSTRACT: Computational Fluid Dynamics is an efficient tool for providing solutions for complex flow dynamics problems in fluid mechanics. Vortex tube happens to be one of such a research field in which this tool can be applied; vortex tube is a device in which energy separation of a compressed fluid is observed. In spite of constant research projects a model for its optimum operation is still in research phase. This research includes investigation of 20 models of vortex tube based on different geometric parameters like cold orifice diameter (d_c), L/d ratio, profile of inlet and number of inlet for a specific diameter tube, from these models an optimum model selected in terms of maximum cold temperature difference (ΔT_c) and cold-mass fraction (μ). This research also includes experimental validation of the optimum model and comparison of performance characteristics of these studied models. Advanced research includes effect of parameters like distance of cold orifice from inlet and back pressure in hot conical valve.

KEYWORDS: Back Pressure in Conical Valve Cold-Mass Fraction (μ), ColdOrifice Diameter (d_c), Cold Temperature Difference (ΔT_c), Computational Fluid Dynamics (CFD), Experimental Validation, L/D Ratio, Profile of Inlet.

I. INTRODUCTION

Vortex tube is a device which can be used as an alternative to refrigeration systems. This device has no moving parts, no chloro-fluoro carbons and uses only compressed gas. This device can also be used effective cooling device with cold end temperatures of some tubes dropping down to minimum of 4 degree Celsius. Compressed stream of gas enters tangentially in a cylinder with conical valve at farther end and cold orifice at the nearer end, this stream is then separated in two streams cold and warm respectively. Cold stream comes out of the cold orifice and hot stream is allowed to escape from the conical hot valve. History of this device can be traced back to 1933, when a French scientist Ranque observed this phenomenon of flow separation in a cylindrical tube. Further in 1940 research was carried out by a German scientist named Hilsch, hence the tube has the name Ranque-Hilsch Vortex Tube (RHVT). Their design was primitive now there have been a lot of improvements in design, but an optimum model is still in research phase.

Computational Fluid Dynamics enlarges the scope of research as it is efficiently solves complex flow equations and also enables us to visualise the flow pattern in terms of graphics and animations. Interaction between the liquids or gases can be simulated according to the boundary conditions provided and observed. Computers are used to do this work. Research carried out in previous work suggests CFD to be a useful tool while investigating vortex tube [2]. This tool can be effectively used for different boundary conditions that can arise in real life situations and visualization of complex experimental setups. Vortex tube is one of such set up which has critical observational restrictions and the flow visualisation option of CFD is useful in this research work.

Current research focuses on obtaining optimum parameters for vortex tube by using CFD and experimental techniques. Objective of this research is to study different geometric parameters, inlet profile, number of inlets, and effect of inlet pressure on vortex tube and obtain optimum set of all these parameters for maximum temperature drop at

cold outlet. This work uses many previous experimental results to provide comparisons with theoretical approximations and thus a base of reference of this research is established.

The first part of research focuses on computational analysis of vortex tube by creating a flow domain to determine the initial optimum cold end temperature difference and geometric parameters like l/d ratio and d_c and validating previous experimental results [1]. Second part of the research involves in more advanced parameters like number of inlet, profile of inlet and intermediate parameters like length of inlet nozzle from cold orifice and pressure at conical valve (hot outlet). Third part of this research was comparison on performance parameters and obtaining an optimum model of the observed results. Final part of this research comprises of manufacturing the obtained optimum tube and comparing the experimental results of this tube with results of Computational analysis.

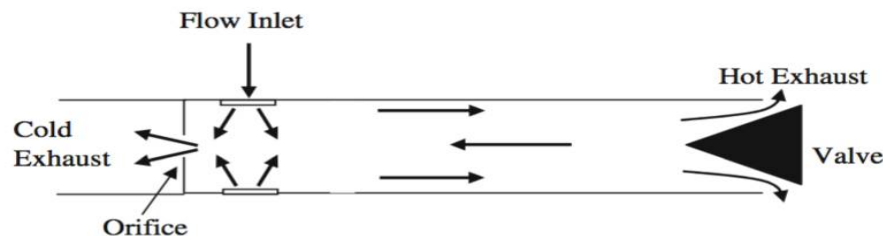


Fig. 1 Schematic Diagram of Vortex Tube

Figure 1 show a schematic diagram of counter flow vortex tube with cold exhaust and hot exhaust at left and right side accordingly with black bold valve and arrows representing the flow direction.

II. RELATED WORK

N. Agrawal, S.S. Naik, Y.P. Gawale [1] has done an experimental investigation on Ranque–Hilsch vortex tube (RHVT). Experiments are conducted on three different vortex tubes with L/D ratios; 12.5, 17.5 and 22.5. Inlet pressure is also varied from 3 to 5 bar in the increment of 1 bar. For the testing, cold mass fraction is varied in the range of 10 to 90% with the step size of 10%. Three different gases — air, nitrogen and CO_2 are tested. It is found that vortex tube performs better with carbon dioxide as working fluid. An experimental investigation is made by D.D. Pawar, B. Sridhar Babu [2] to determine optimum geometry of Ranque-Hilsch vortex tube by increasing number of nozzle and supply pressure. Numerical investigation is carried out by Hitesh R. Thakare, Aniket Monde, Bhushan S. Patil, and A. D. Parekh [3] of flow characteristics in counter-flow vortex tube. A vortex tube with internal diameter 11.4 mm and working length of 106 mm was used. Air was admitted as working fluid of vortex tube. The cold end diameter was fixed to be 6.2 mm. Maximum cooling power separation is obtained for cold mass fraction in the range of 0.6 to 0.7, unlike maximum cold end temperature separation. Hitesh R. Thakare, Aniket Monde, A. D. Parekh [4] performed three-dimensional (3-D) computational fluid dynamics (CFD) analysis of counter-flow vortex tube. CFD simulation of vortex tube was attempted in order to understand nature of flow physics inside the tube. K.K. Zin, A. Hansske and F. Ziegler [5] have done study on modelling and optimization of vortex tube with the help of computational fluid dynamic analysis. CFD analyses were carried out for a 32 mm diameter vortex tube with L/D of 5, 10, 15 and 20 verify the existence of a secondary circulation flow, a stagnation point. And also with the varying amount of cold fraction 0.1, 0.2, 0.35, 0.5, 0.7 has been varied to investigate the variation of static and total pressure as well as the velocity components of the particle. A three-dimensional numerical model of the Ranque-Hilsch vortex has been established to analyse the flow inside the tube. The studies have confirmed the presence of a secondary flow. Nader pourmahmoud, Amir hassanzadeh, Omidmoutaby, and Abdolrezabramo [6] have worked on Computational Fluid Dynamics Analysis of Helical Nozzles Effects on the Energy Separation in a Vortex Tube. Three kinds of nozzles set include of 3 and 6 straight and 3 helical nozzles have been investigated and their principal effects as cold temperature difference were compared. The total temperature separations (hot and cold exit) predicted by the CFD model of 6 straight nozzles were found to be in a good agreement.

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 5, Issue 5, May 2016

III. VORTEX TUBE DESIGN

A. Vortex tube terminology:

Schematic diagram of vortex tube is shown in figure 2 and consists of following parts,

- (a) Nozzle (Inlet)
- (b) Cold orifice (cold outlet)
- (c) Hot valve (hot outlet)
- (d) Tube

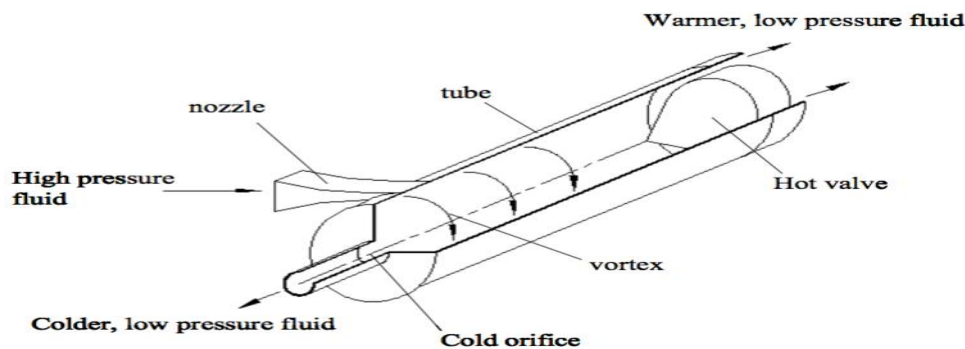


Fig. 2 Vortex Tube Terminologies

B. Design constraints

Geometrical parameters play an important Role in performance characteristics of vortex tube. Many researchers have obtained an optimum model, for the same but still follow a certain amount of ambiguity. However following restrictions are suggested while designing a vortex tube,

- 1) For obtaining maximum temperature difference l/d ratio of the tube needs to be kept in the range of $10 \leq l/d \leq 30$.
- 2) For maximum efficiency and temperature difference at cold end d_c/d ratio should be in range of $0.4 \leq d_c/d \leq 0.6$.
- 3) Cold mass fraction for maximum efficiency should be between $0.3 \leq \mu \leq 0.6$.
- 4) Inlet pressures are kept in the range of 5 bar to 6.5 bar.

IV. COMPUTATIONAL FLUID DYNAMICS ANALYSIS

A. Governing equations in computational fluid dynamics:

The equations governing the fluid motion are the three fundamental principles of mass, momentum, and energy conservation.

- Continuity $(\partial \rho / \partial t) + \nabla \cdot (\rho \mathbf{V}) = 0$ (1)

- Momentum $\rho (d\mathbf{V}/dt) = \nabla \tau_{ij} - \nabla p + \mathbf{F}$ (2)

- Energy $\rho (de/dt) + p (\nabla \cdot \mathbf{V}) = (\partial Q/\partial t) - (\nabla \cdot \mathbf{q}) + \Phi$ (3)

Where ρ is the fluid density, \mathbf{V} is the fluid velocity vector, τ_{ij} is the viscous stress tensor, p is pressure, \mathbf{F} is the body forces, e is the internal energy, Q is the heat source term, t is time, Φ is the dissipation term, and $\nabla \cdot \mathbf{q}$ is the heat loss by conduction. Fourier's law for heat transfer by conduction can be used to describe \mathbf{q} as: $\mathbf{q} = -k\nabla T$.

B. Mesh parameter:

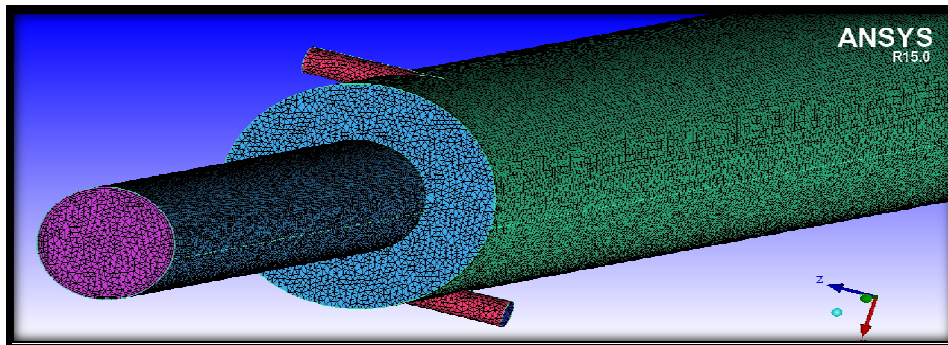


Fig. 3 Surface Mesh of Vortex Tube

Figure 3 shows the mesh of double tangential inlet vortex tube made in ICEM CFD with enlarged view of the inlet and cold orifice part of the model. Number of Tetra Elements: 1100400, Quality: 0.4, Volume Mesh: Tetragonal Robust Octree with prism layers. Tetra mesh gave desired results, but in this case hexahedral mesh can also give optimum results.

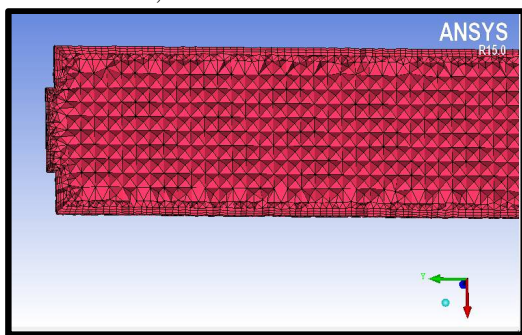


Fig. 4 Volume Mesh of Cold Orifice

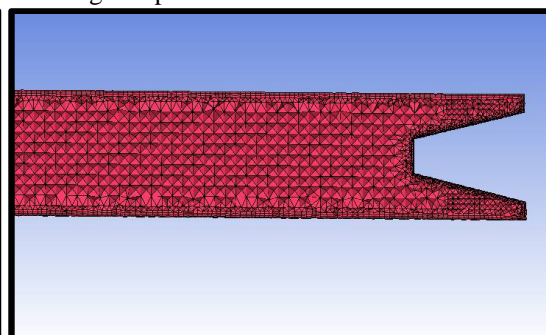


Fig. 5 Volume Mesh of Hot Orifice

Figure 4 shows the enlarged view of cut plane for volume mesh at the cold outlet and figure 5 shows the cut plane of volume mesh for hot outlet these both figures show a fine prism layer of mesh at the boundary or at walls.

C. Boundary conditions:

Table I. Boundary Conditions for all Types of Vortex Tube

Sr. no.	Parameters	Value
1	Cold Pressure Outlet	Type- Pressure Outlet, Gauge Pressure = 0, Hydraulic Dia.= 9 mm
2	Hot Pressure Outlet	Type- Pressure Outlet, Gauge Pressure = 0, Hydraulic Dia.= 0.9 mm
3	Mass flow inlet	Super-sonic Pressure=5.6 bar, Hydraulic Dia.=2 mm
4.	Inlet Pressure	Type- Mass Flow-rate, 5 bar
5.	Inlet Temperature	300 K
6.	Turbulence Model	K-epsilon, realizable, simple.
7.	Turbulent Wall Function	No slip

These boundary conditions recreate the exact experimental set up for the vortex tube. These conditions also included heat flux through wall as zero and enhanced wall treatment.

D. Turbulent models and solver used:

Turbulent models which were suitable for mixing or separating flow were k-epsilon and k-omega, of these models k-epsilon model gave desired results. Being a two equation turbulent model it was used and also its attachments with realizable and standard showed optimum results while RNG model showed divergence. Solver used was solver with SIMPLE pressure velocity coupled solver, initially if divergence was detected in the models COUPLED solver was used for initial iteration and then the analysis was continued on SIMPLE solver.

V. COMPUTATIONAL RESULTS AND DISCUSSIONS

In this chapter the CFD results are summarised and relative comparison has been presented. This section deals with different inlet parameters are compared with maximum temperature drop at cold outlet at different inlet pressures.

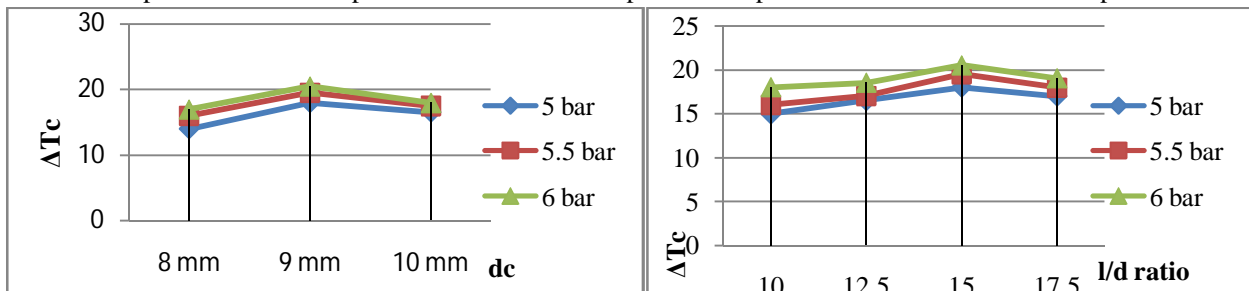


Fig. 6 d_c Vs ΔT_c of Computational Results

Fig. 7 l/d ratio Vs ΔT_c of Computational Results

Figure 6 shows the graph of cold orifice diameter versus cold end orifice temperature difference with inlet, in this graph it is observed that maximum temperature difference at cold outlet is achieved at diameter of 9 mm which is 0.5 times tube diameter. Figure 7 shows the variation in cold end temperature difference with variation in l/d ratio of the tube, the maximum temperature difference is seen to be achieved at 15 l/d ratio.

Table II. Performance Characteristics of Different Vortex Tube

Sr. no.	Types of Vortex Tubes	Isentropic Efficiency %	COP
1	Single Inlet Spiral	11.8	0.07
2	Single Inlet Tangential	15.53	0.09
3	Double Inlet Spiral	10.89	0.06
4	Double Inlet Tangential	17.26	0.1
5	Triple Inlet Spiral	14.53	0.05
6	Triple Inlet Tangential	19.07	0.09

Table II contains the values of COP and Isentropic efficiency obtained for each inlet profile and number of inlet as computational results. Following are the graphs of COP and Isentropic efficiency with change in inlet type

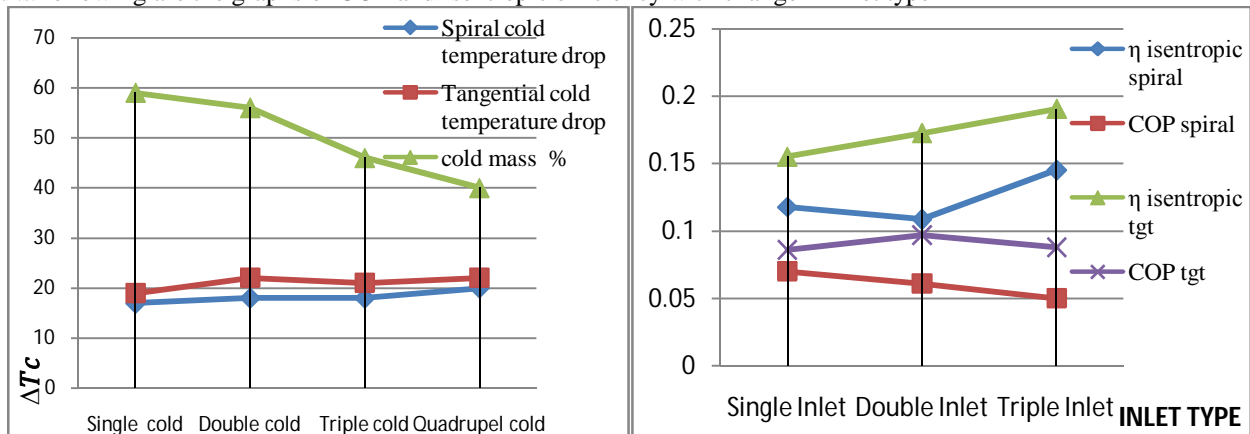


Fig. 8 No. of Inlet Vs ΔT_c of Computational Results Fig. 9 η Vs COP isentropic vs. No. of inlet of CFD Results

Figure 8 and Figure 9 show the comparison in performance characteristics in number of inlets for the tube of different inlet profiles like tangential or straight and single double and triple inlet.

Out of these models double inlet tangential showed optimum results with maximum temperature difference greater efficiency and better COP than rest of the models.

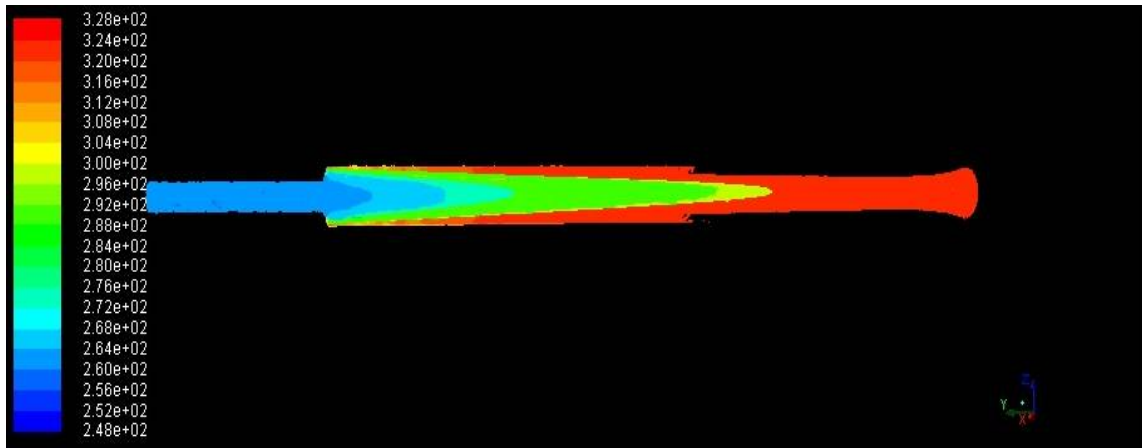


Fig. 10 Internal total temperature contours of optimized model

Figure 10 represents the model of double tangential inlet with temperature contours of the maximum temperature difference obtained along with optimum performance parameters. A distinct cold core can be seen with various temperature profiles in which blue represents cold and red represents hot.

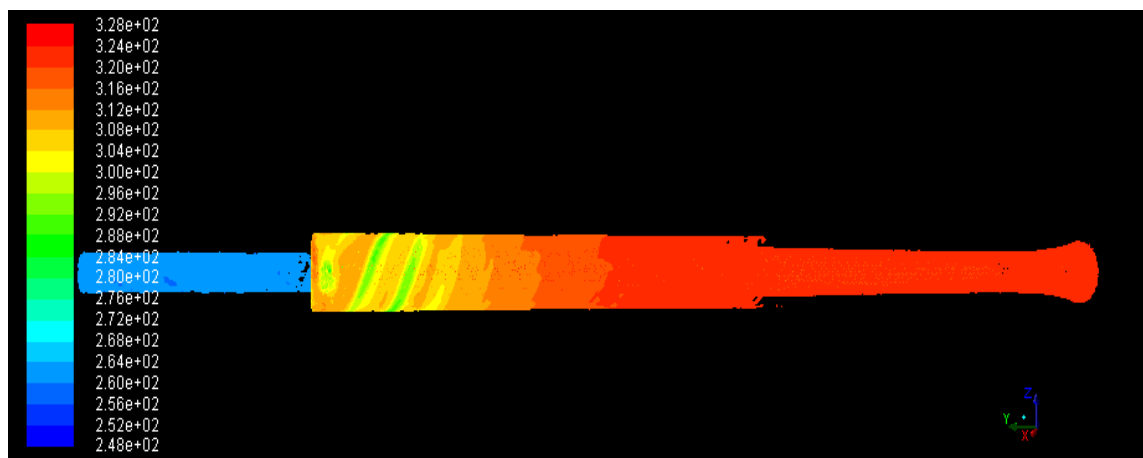


Fig. 11 Surface temperature values of optimum model

Figure 11 depicts the temperature distribution along the surface of the tube, distinct cold and hold parts can be seen in this figure blue being the cold outlet and red being the hot outlet.

VI. EXPERIMENTAL SETUP

In order to create a two tangential inlet for a vortex tube it was necessary to build a chamber where compressed air would gush in before entering tangentially in the tube periphery. A chamber was thus made which had two tangential inlets inside the tube and a space to gush compressed air in; this chamber was manufactured from PVC and is shown in the figure 12.

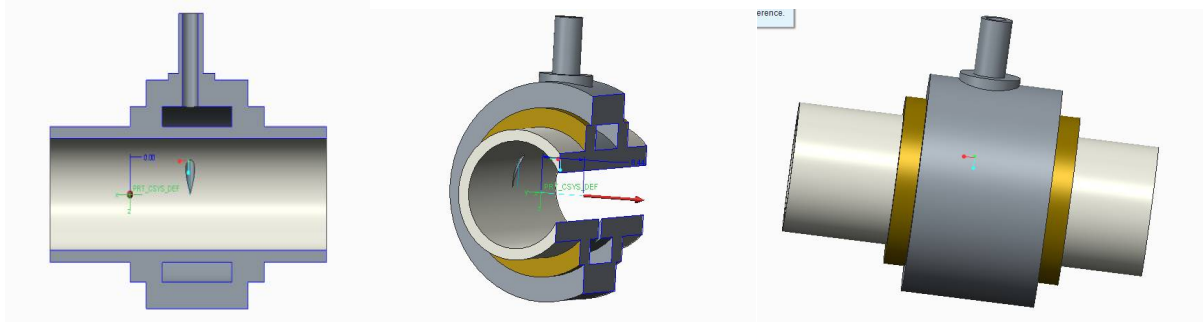


Fig. 12.1 Fig. 12.2 Fig. 12.3

Figures 12.1, 12.2, 12.3 contain views of the vortex chamber created in creo parametric which represent the construction of vortex chamber and its various views from sectional to full solid view.



Fig 13.1 vortex-chamber Fig 13.2 Hot valve assembly Fig 13.3 Hot valve Fig 13.4 cold orifice

Figure 13.1 shows the vortex chamber manufactured from PVC, figure 13.2 shows the assembly of hot outlet valve, figure 13.3 shows the hot outlet valve which was manufactured on lather from aluminum and holes were perforated on the casing for hot air to exit and figure 13.4 shows the cold outlet which was made from wood on a wood turning machine and drilled with a through hole.

VII. COMPARISON OF RESULTS OF OPTIMUM TUBE

This section deals with comparison of results of CFD and experimental model.

Table III Comparison of Experimental and Computational results (Validation)

Sr. No.	Parameters	Experimental value	Computational value
1	Cold end temperature	282 k	281 k
2	Hot end temperature	320 k	322 k
3	Cold mass fraction	56%	56%

Table III Shows the comparison between the computational and experimental results of optimized vortex tube followed by performance characteristics of both results.

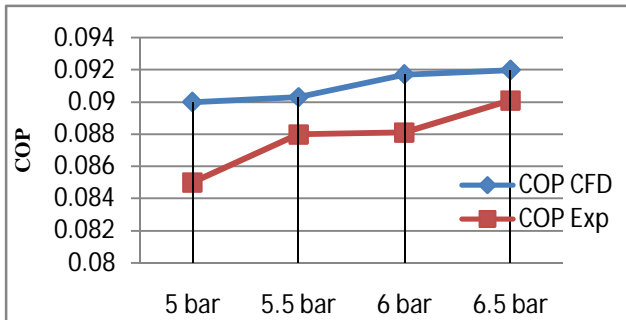


Fig. 14 Inlet Pressure in Vs η Isentropic

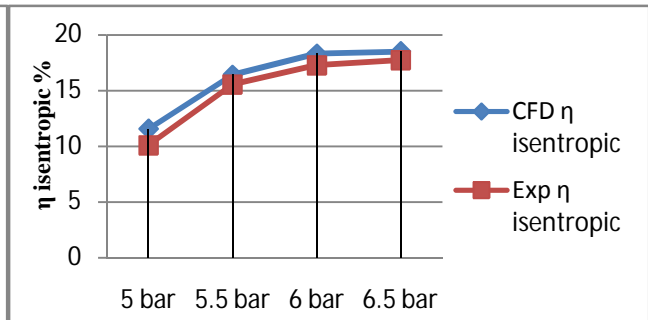


Fig. 15 Inlet Pressure in Vs COP

Figure 14 and Figure 15 show with increase in inlet pressure both COP and Isentropic efficiency of experimental as well as computational models increases with no notable fluctuation; hence results are validated when compared with increase in inlet pressure.

VIII. EFFECT OF CHANGE OF PARAMETERS ON OPTIMIZED TUBE

This chapter discusses effect of change in parameters of optimized tube which were not considered in previous research.

a) Effect of change in distance between cold orifice diameter and inlet:

Increase in distance of inlet from cold orifice causes the flow to distort and gives rise to a phenomenon called trapping.

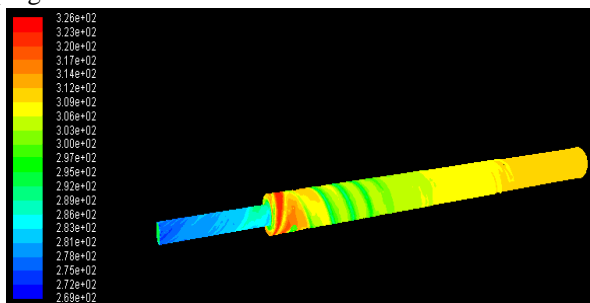


Fig. 16 Temperature Spike (Trapping)

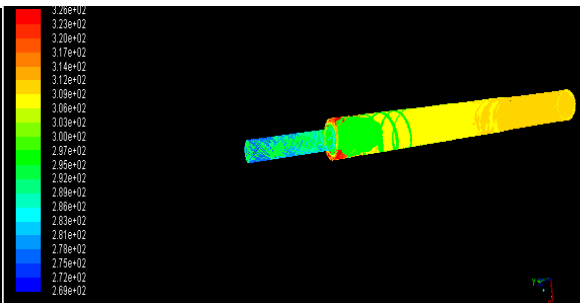


Fig. 17 Internal Flow Distortion Due To Spike

Shown in figure 16 the red colored spike in vortex tube inlet side gives the contour of trapped air which has temperature greater than the inlet air, this causes the cold temperature difference to decrease and this also decreases the performance of the tube. Figure 17 shows the cross-sectional temperature contour of trapping.

b) Effect of increase in back pressure of hot conical valve:

Increase in back pressure of hot conical valve can be caused due to insufficient space of the hot gas to escape this causes a phenomenon called choking. This decreases the cold end temperature by causing flow distortion due to choking.

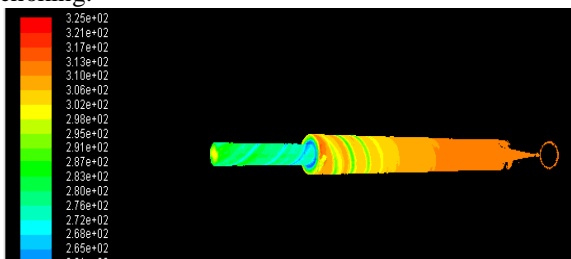


Fig. 18 Choking Phenomenon

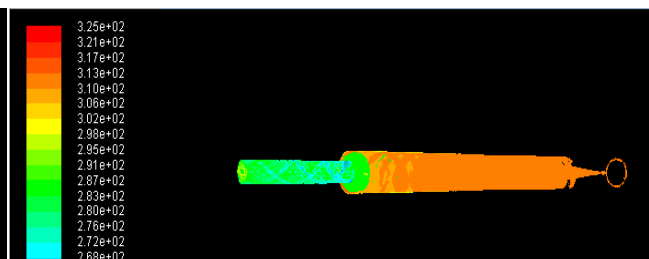


Fig. 19 Internal Flow Distortion Due To Choking

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 5, Issue 5, May 2016

Fig 18 and 19 show the choking phenomenon. Orange coloured hot outlet represents less temperature obtained at hot end and distorted cross-sectional flow respectively. Towards the end of hot valve a diminishing flow is observed this is due to excess back pressure at the hot outlet.

IX. CONCLUSION

After observing above results it can be concluded that Computational Fluid Dynamics is an efficient tool for research in field of vortex tube. This research can be concluded with an optimum model of vortex tube with double tangential inlet for 18 mm diameter tube, temperature difference (ΔT_c) was observed to be 20 kelvin and cold-mass fraction (μ) was 56%. It was also observed that tube gave optimum results for cold orifice diameter of 9 mm with is 0.5 times tube diameter; l/d ratio of 15 gave optimum results. Experimental validation of the optimum tube was close to CFD results and outcome was acceptable. Even after making a tube of optimum geometry certain phenomenon can affect the performance of the tube like choking and trapping, steps should be taken to avoid these actions in the vortex tube, minimum distance between cold orifice and optimum space for hot air to escape can be effective alternatives.

REFERENCES

- [1] N. Agrawal, S.S. Naik, Y.P. Gawale, "Experimental Investigation of Vortex Tube Using Natural Substances.", International Communications in Heat and Mass Transfer [online], Pp. 51–55, 2014.
- [2] D.D. Pawar, B. Sridhar Babu, "Computational Fluid Dynamics and Experimental Analysis for Optimum Geometry of Vortex Tube", International Journal of Innovative Research in Science, Engineering and Technology [online] Volume 3, Issue 8, 2014.
- [3] Hitesh R. Thakare, Aniket Monde, Bhushan S. Patil, A. D. Parekh, "Numerical Investigation of Flow Characteristics in Counter-Flow Vortex Tube", International Conference on Computational Heat and Mass Transfer, Procedia Engineering 127, Pp. 170 – 176, 2015.
- [4] Hitesh R. Thakare, Aniket Monde, A. D. Parekh, "3-D CFD Analysis of Vortex Tube", International Journal of Modern Trends in Engineering and Research, Volume 2, Issue 7, 2015.
- [5] K.K. Zin, A. Hansske, F. Ziegler, "Modeling and Optimization of the Vortex Tube with Computational Fluid Dynamic Analysis", Energy Research Journal, Volume 1, Issue 2, Pp.193-196, 2010.
- [6] Nader Pourmahmoud, Amir Hassan Zadeh, Omid Moutaby, Abdolreza Bramo, "Computational Fluid Dynamics Analysis of Helical Nozzles Effects On the Energy Separation In A Vortex Tube", Thermal Science, Vol. 16, No. 1, Pp. 151-166, 2012.
- [7] Upendra Behera, P.J. Paul, S. Kasthuriengan, "CFD analysis and experimental investigations towards optimizing the parameters of Ranque-Hilsch vortex tube", International Journal of Heat and Mass Transfer, 2004.
- [8] Kirmaci Volkan, "Exergy analysis and performance of a counter flow Ranque-Hilsch vortex tube having various nozzle numbers at different inlet pressures of oxygen and air", International Journal of refrigeration, Pp.1626-1633, 2009
- [9] Nimbalkar S., Muller M. R., "An experimental investigation of the optimum geometry for the cold end orifice of a vortex tube", Appl. Therm. Engineering, Pp. 509–514, 2009.
- [10] Bramo, A. R., Pourmahmoud N., "CFD simulation of length to diameter ratio effects on the energy separation in a vortex tube", IJRRAS, Pp 1-16, 2009.
- [11] M. Yilmaz, M. Kaya, S. Karagoz, S. Erdogan, "A Review on Design Criteria for Vortex Tubes", Heat Mass Transfer 45, Pp. 613–632, 2009.
- [12] Y.T. Wu, Y. Ding, Y.B. Ji, C.F. Ma, M.C. Ge., "Modification and Experimental Research On Vortex Tube", Int. J. Refrig., 1042–1049, 2010.