CFD Simulation and Experimental Validation to Obtain Optimized Vortex Tube

Mr.Ravindra Baban Zanje[1], Prof.S.K.Bhosale[2]

[1]PG Student, Dept. of Mechanical Engineering, KJ's Trinity College of Engineering and Research, Maharashtra, India
[2]Prof. of Dept. of Mechanical Engineering, KJ's Trinity College of Engineering and Research, Maharashtra, India

---***---

Abstract- *The vortex tube (VT) is a mechanical instrument that can deliver warming and cooling. The Vortex Tube cooler is a gadget that creates cold and hot gas from packed gas. It contains at least one bay spouts, a vortex chamber, a cool end hole, a hot-end control valve and a cylinder. At the point when high-pressure gas (10 bar) is extraneously infused into the vortex chamber by means of the delta spouts, a whirling stream is made inside the vortex chamber. At the point when the gas whirls to the focal point of the chamber, it is extended and cooled. In the vortex chamber, some portion of the gas whirls to the hot end, and another part exists through the virus fumes straightforwardly. Some portion of the gas in the vortex tube turns around for hub part of the speed and move from the hot end to the virus end. In the trial arrangement, the temperatures inside the funnel at the bay, hot outlet and cold outlet were estimated by thermocouples. In this work, a 3D CFD model is first evolved in CATIA software and reenacted in ANSYS. In present research 3 model of vortex tube are concentrated to acquire streamlined vortex tube to be specific single channel, 4 inlet and 6 inlets utilizing CFD simulation in ANSYS software. Trial assembling of improved structure and approval of results with CFD results.*

Keywords—ANSYS Fluent, CATIA, CFD, Vortex tube

1. INTRODUCTION

The vortex tube is a basic instrument that is equipped for isolating a high-pressure stream into two lower pressure streams, altogether more sweltering and cooler than the delta liquid. A run of the mill counter-stream vortex tube is an empty cylinder, which has at least one extraneous bay spouts, a focal cold spout and a fringe hot exit. At the point when compacted gas is conceded digressively into the cylinder, solid twirl stream is shaped and in the meantime temperature partition happens, specifically, liquid in the focal area chills off and liquid in the fringe locale heats up. The cool liquid leaves the focal opening close to the passage spout, while the hot liquid releases the outskirts at the furthest finish of the cylinder. A vortex tube incorporates various parts, for example, at least one bay spouts, a vortex-chamber, a virus end hole, a throttle valve that is situated toward the finish of primary cylinder and a working cylinder. At the point when constrained liquid is gone into the vortex-chamber extraneously through the spouts, a solid rotational stream field is made. At the point when the liquid extraneously twirls to the focal point of the vortex tube it is extended and cooled. After event of the vitality partition in the vortex tube the forced bay liquid stream was separated into two distinctive liquid streams including hot and cold fumes liquids.

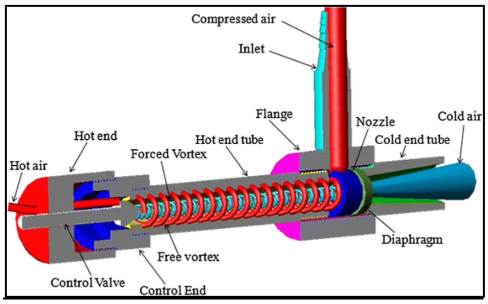


Fig.1 Sectional view of vortex tube

The "cool exit or cold opening" is situated at close to the bay spouts and at the opposite side of the working cylinder there is a variable stream limitation part in particular the control or throttle valve which decides the mass stream pace of hot exit. A percent of the compacted gas escapes through the valve toward the finish of the working cylinder as hot stream and the rest of the gas returns in an internal twirl stream and leaves through the virus leave opening. Opening the throttle valve decreases the chilly wind current and shutting the valve expands the virus mass stream proportion.

2.LITERATURE REVIEW

Sirajuddin Syed et al. [1], in this paper it presents the examination which includes the inlet energy considered for separation of energy.Three-dimensional computational liquid unique reenactment in vortex cylinder to examinations the vitality detachment wonders in various cases is performed. They utilize the diverse working medium, for example, hydrogen and air which have variety in explicit warmth. In the two cases they modify the channel mass stream rate and keep up the energy at the inlet. Vortex tube with hydrogen as working medium yields a temperature partition of 8 K lower than air as working medium. Further examinations on vortex tube with hydrogen as a working liquid is investigated at various bay temperatures comparative with the room temperature. It is reasoned that the outcome that vortex tube with hydrogen at a channel temperature of 400 K gives better temperature detachment when contrasted with other delta temperatures considered for the investigation.

Kiran Dattatraya Devade et al. [2], in this paper it presents study to the stream conduct of vortex tube. This is a vitality detachment instrument that isolates packed gas stream into a low and a high temperature stream. Research is accomplished for the stream conduct inside the vortex tube for various generally utilized liquids with changed properties like Air, He, N_2, CO_2 and NH_3. Recreation is done for various violent models, standard k-ε, and Realizable k-ε and RNG k-ε. They utilize the Realizable k-ε model for investigation. They examine the stream conduct of gases with shifted multi-nuclear number is and contrasted and writing. Result is discovered that the impact on temperature for N_2 is better, trailed by He, CO_2, Air and NH3. Vitality partition for N_2 is 46 % higher than every single different gas.

A. Aghagoli et al. [3], in this paper it presents on CO_2 as a working medium. To reproduce the progression of CO_2 inside a vortex tube 3D CFD model is first evolved and afterward it is approved with distributed test information. The accepted k – ε choppiness model uses Structured Hexahedral hubs created in ANSYS Meshing. Just 1/6 of the geometry is required because of the evenness of the VT. They consolidate the approved CFD model with a thermodynamic model of the VT to do a parametric report, where the gulf pressure (550 kPa to 1300 kPa) and cold mass portion (0.2–0.9) are the picked parameters. Here the energy partition is talked about as far as the hot exit and cold leave temperature contrasts, both concerning the VT inlet temperature. From the numerical aftereffects of the investigation it is show that the variety of the virus mass part from 0.2 to 0.9, for a fixed gulf weight of 1300 kPa, causes the hot leave temperature contrast to ascend from 10 °C to 78.9 °C, while the cold temperature distinction tumbles from 44.2 °C to 9.7 °C.

H.M. Skye et al. [4], in this paper it present examination between the presentation anticipated by a computational fluid dynamic (CFD) model and test estimations taken utilizing an industrially accessible vortex tube. The CFD model is a two-dimensional (2D) consistent axisymmetric model (with twirl) that uses both the norm and renormalization gathering (RNG) k-epsilon choppiness models. While CFD has been utilized beforehand to comprehend the liquid practices interior to the vortex tube, it has not been applied as a prescient model of the vortex tube so as to build up a plan instrument that can be utilized with certainty over a scope of working conditions and geometries. The fundamental target of this paper is effective utilization of CFD which is utilized as incredible asset that can be utilized to enhance vortex tube configuration just as survey its utility with regards to new applications.

Jacob Leachman et al. [5], in present article it proposes Ranque–Hilsch vortex tubes are utilized to create cooling when a packed air source is promptly accessible. When contrasted with customary cooling advances, vortex-tube gadgets are a lot less difficult and they don't have moving parts. In this paper creator do the examination numerically for vortex tubes with cyclonic-type expansions of the vortex chambers. To simulate a run of the mill counter-stream vortex tube working with pretence the computational liquid elements programming STAR-CCM + is used. The investigation is accomplished for structure of vortex tube for the effective high limit. It is inferred that the outcome that moderate size vortex chamber with expansion exhibited the best among the considered arrangements. This variation of the vortex tube was additionally anticipated to perform better than the first framework by about 15% in a scope of cold-stream portions.

3. PROBLEM STATEMENT

Nowadays, it is observed that modification in vortex tube is increasing day by day to enhance the cold temperature at cold side so, in present research different inlet geometry are presented to obtain optimized geometry for most efficient vortex tube to be used for multipurpose applications.

4. OBJECTIVES

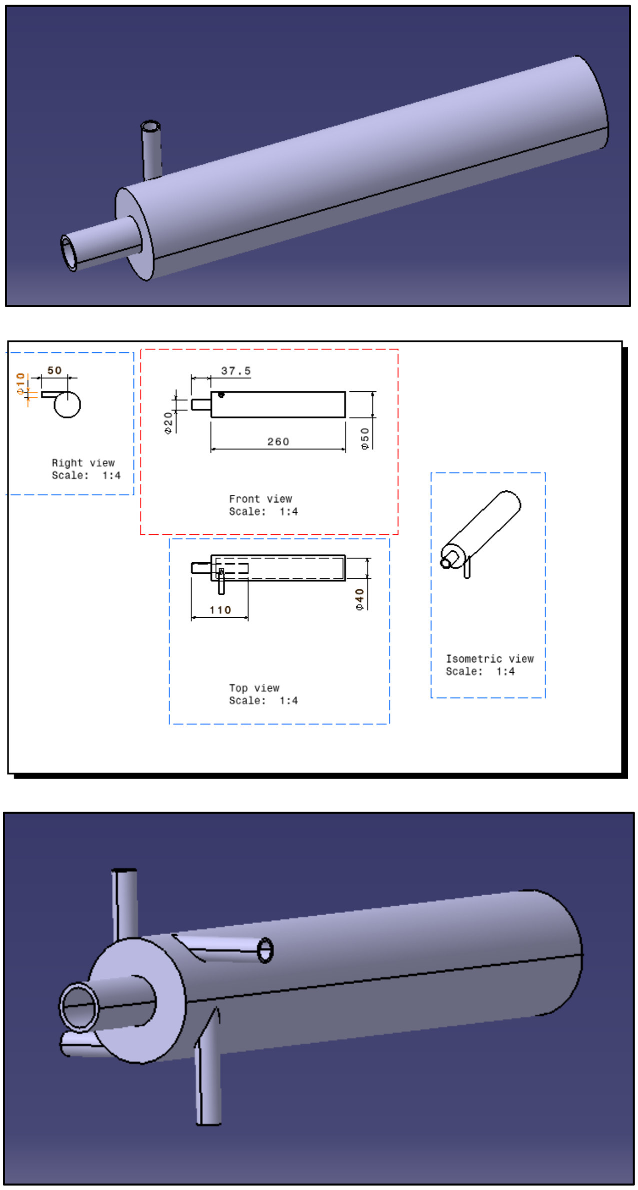
1. Design of single, 4 inlet and 6 inlet geometry in CATIA software to obtain optimized vortex tube geometry.
2. CFD simulation using ANSYS software for different geometries with compressed pressurized air at inlet nozzle to determine the hot and cold temperature at both sides.
3. In the experiments, inlet nozzles angle generators were changed from the largest angle to the smallest angle in the range.
4. Experimental setup of modified vortex tube to obtain hot and cold temperature sensing using thermocouple to determine temperature across it.
5. Comparison of experimental and CFD simulation results to check the parameters affecting the performance.

5. METHODOLOGY

Step 1:- Initially research paper are studied to find out research gap for project then necessary parameters are studied in detail. After going through these papers, we learnt about optimum design of vortex tube.
Step2:- Research gap is studied to understand new objectives for project.
Step 3:- After deciding the components, the 3D Model and drafting will be done with the help of CATIA software.

Step 4:- Computational Fluid Dynamics (CFD) simulations of vortex tube with single input nozzle and multiple input nozzle will be done with the help of ANSYS Fluent software.
Step 5:- The manufacturing of optimized model will be done, after that experimental reading are note down with the help of thermocouple.
Step 6:- Comparative analysis between the experimental &CFD result & then the result & conclusion will be drawn

6. DESIGN OF VORTEX TUBE



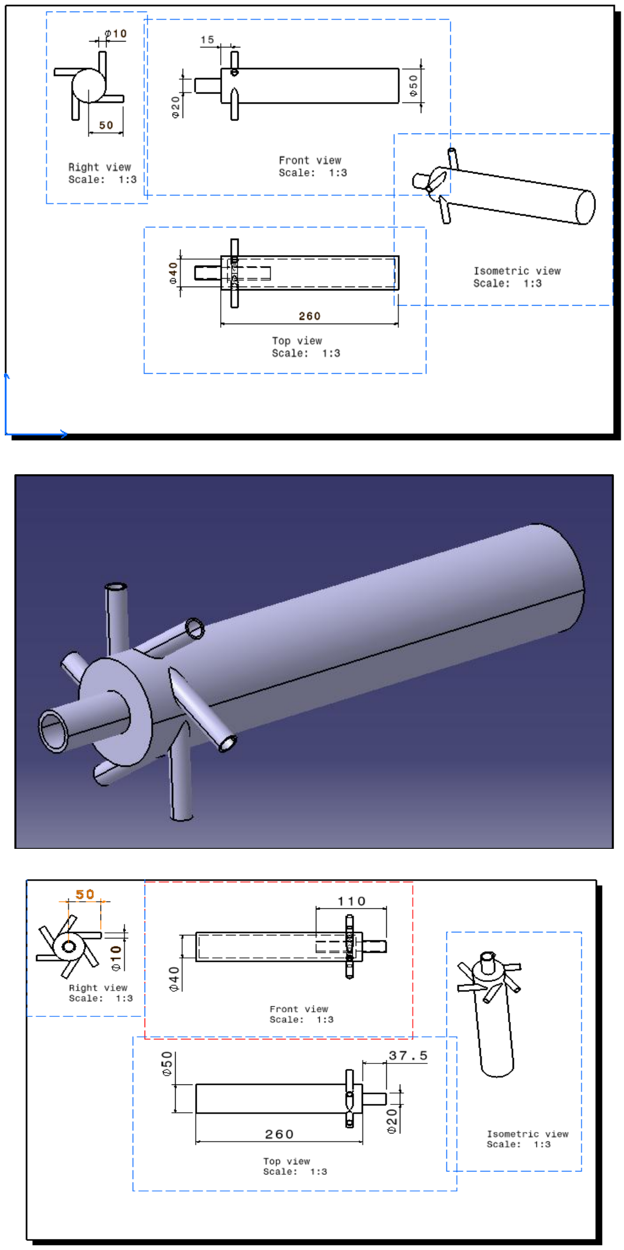


Fig.2 CATIA and drafting for different inlet geometry for vortex tube

Computational fluid dynamics (CFD) is a branch of fluid mechanics that uses numerical analysis and data structures to analyze and solve problems that involve fluid flows.

CFD is now recognized to be a part of the computer-aided engineering (CAE) spectrum of tools used extensively today in all industries, and its approach to modelling fluid flow phenomena allows equipment designers and technical analysts to have the power of a virtual wind tunnel on their desktop computer.

6.1 CFD PROCEDURE

- In CFD simulation bounding box is created across blade profile for simulation of velocity and pressure distribution across surface of blade.
- Fine meshing is performed for CFD simulation.
- Named selection is performed in CFD to define air inlet, outlet and blade surface.
- In general box model gravity is defined in perpendicular direction and energy is kept on to perform conservation of mass, momentum and energy equation to solve.
- In viscous model k epsilon, realizable and standard wall function is selected to maintain turbulence flow.
- Inlet pressure is defined as 10 bar.
- Hybrid initialization is performed.
- 100 number of iterations is considered.

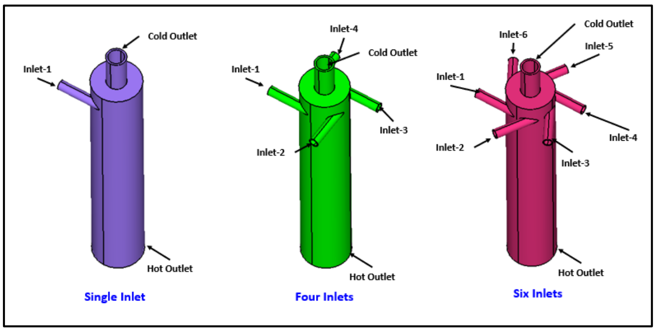
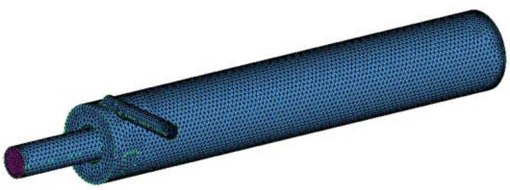


Fig.3 Geometry imported in ANSYS

Mesh

ANSYS Meshing may be a all-purpose, intelligent, automated high-performance product. It produces the foremost acceptable mesh for correct, economical metaphysics solutions. A mesh well matched for a selected analysis may be generated with one click for all elements in a very model. Full controls over the options accustomed generate the mesh are accessible for the skilled user who needs to fine-tune it. The ability of parallel processing is automatically accustomed reduce the time you have got to wait for mesh generation



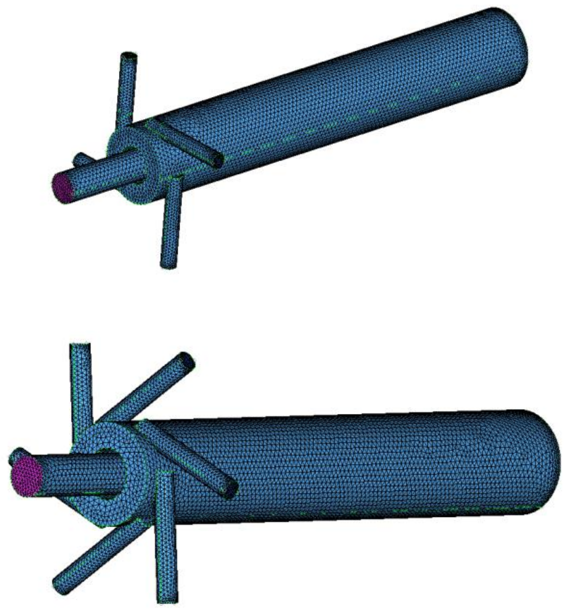


Fig.4 Geometry and meshing of blade

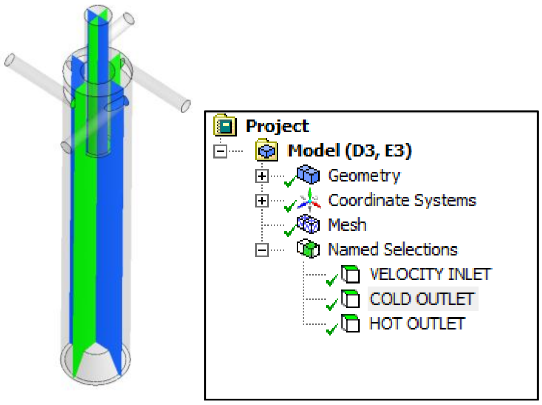


Fig.5 Named selection details and mid plane

Boundary Condition

A boundary condition for the model is that the setting of a well-known value for a displacement or an associated load. For a specific node you'll be able to set either the load or the displacement but not each. The main kinds of loading obtainable in FEA include force, pressure and temperature. These may be applied to points, surfaces, edges, nodes and components or remotely offset from a feature.

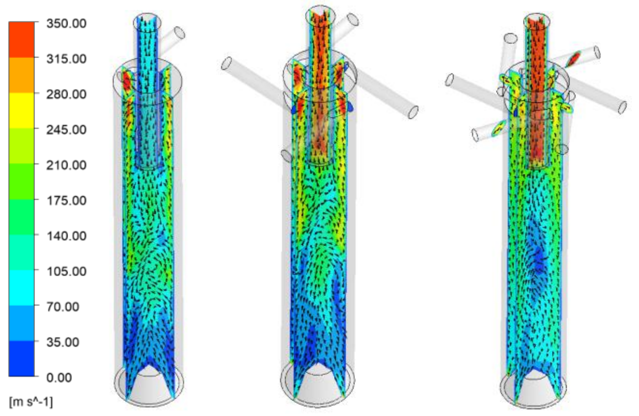


Fig.6 Contour of Velocity Magnitude on Mid Plane (Velocity Range 0-350m/s)

The flow rate inside pipe increases with increase in number of inlets. The increased flow rate is also increasing flow rate on cold outlet as well as on hot outlet.

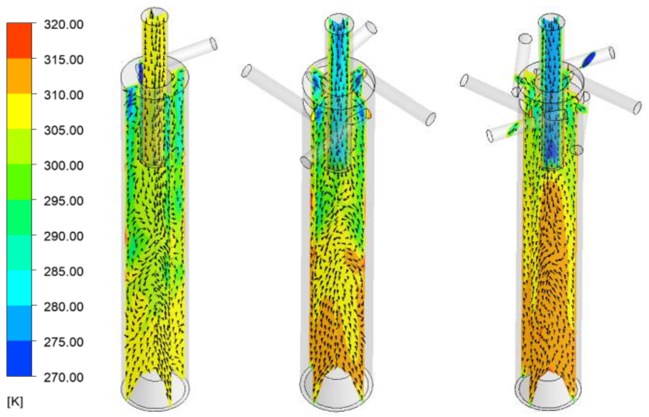


Fig.7 Contour of Temperature on Mid Plane (Temperature Range 270k - 320k)

As the flow is increasing with increase in number of inlets, the temperature on hot outlet is increasing whereas temperature on cold outlet is decreasing further. The increase in temperature difference between hot & cold outlet is happening because of the increase in sudden contraction & expansion inside pipe which is caused due to high swirling of the flow. The swirling of the increases with increase in flow rate.

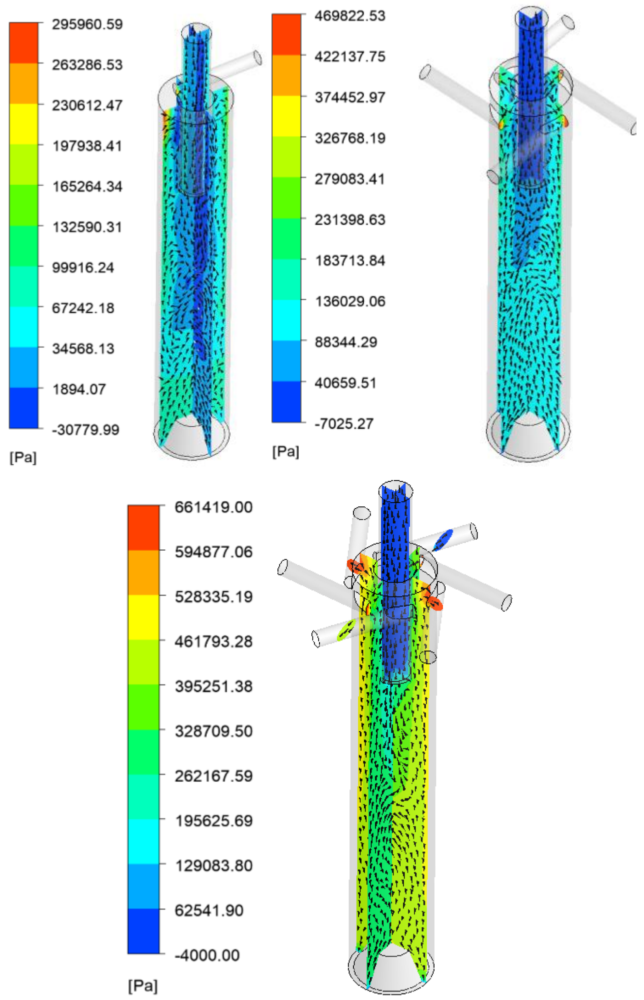


Fig.8 Contour of Pressure on Mid Plane (Auto Range)

The pressure condition inside pipe is increasing with increase in number of inlets. The pressure drop is increasing with increase in number of inlets.

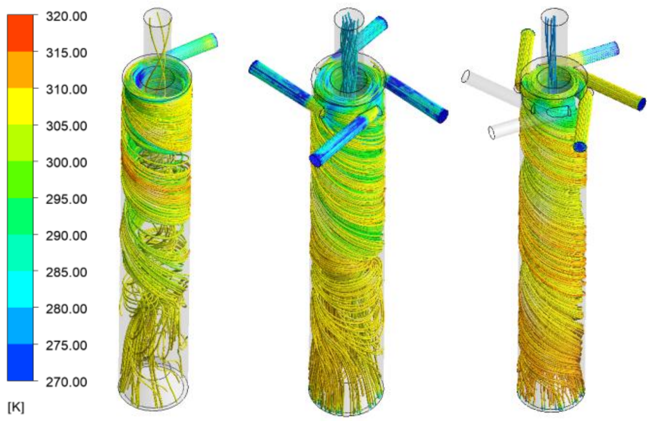


Fig.9 Streamlines Coloured by Temperature Forward from Inlet (Temperature Range 270k - 320k)

As the flow is increasing with increase in number of inlets, the temperature on hot outlet is increasing whereas temperature on cold outlet is decreasing further. The increasing in temperature difference between hot & cold outlet is happening because of the increase in sudden contraction & expansion inside pipe which is caused due to high swirling of the flow. The swirling of the increases with increase in flow rate.

7. EXPERIMENTAL PROCEDURE

- Initially design is designed in CATIA software and fixture is assemble to hold the component.
- Pipe of specified dimension are selected and welded together to form 6 inlet vortex tube.
- Along with end hose is machined using lathe matching i. e. turning operation.
- Compressor is used to transfer compressed air around 10 bars as inlet in 6 inlet pipes combined from a single pipe to distribute it uniformly in each inlets.
- Thermocouple is used to measure cold temperature at cold side inserted it junction in pipe.

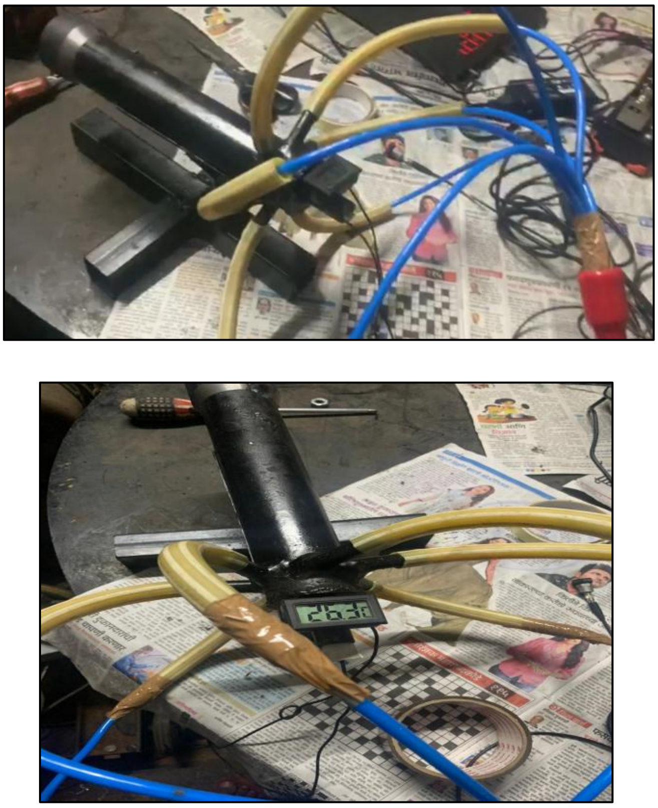


Fig.10 Experimental testing

It is observed that at cold side temperature is around 26^0 C or 299 K.

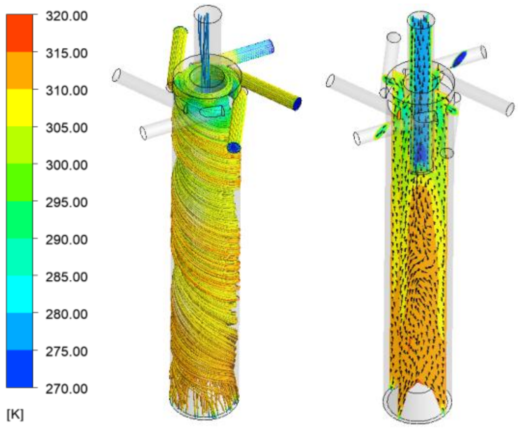


Fig.11 CFD temperature distribution

- During CFD simulation temperature at cold side is observed around average 295 K.
- So, it is validated using experimental result that temperature is around 299 K and CFD simulation is around 295 K.

8. CONCLUSIONS

- In present investigation design of vortex tube have been performed with standards from research papers and CFD simulation for compressed air up to 10 bar is simulated.
- The flow rate inside pipe increases with increase in number of inlets.
- As the flow is increasing with increase in number of inlets, the temperature on hot outlet is increasing and temperature on cold outlet is decreasing further.
- The increase in temperature difference between hot & cold outlet is happening because of the increase in sudden contraction & expansion inside pipe which is caused due to high swirling of the flow. The swirling of the increases with increase in flow rate.
- The pressure is increasing inside pipe and also, the pressure drop across pipe is increasing with increase in number of inlets.
- The increase in number of inlets is improving the performance of the vortex breaker but note that pressure drop limit might be the concern for increase in number of inlets.
- During CFD simulation temperature at cold side is observed around average 295 K.
- So, it is validated using experimental result that temperature is around 299 K and CFD simulation is around 295 K.

REFERENCES

1) Sirajuddin Syed, Manimaran Renganathan, "Numerical investigations on flow characteristics and energy separation in a Ranque Hilsch vortex tube with hydrogen as working medium", International journal of hydrogen energy (2019)
2) Fachun Liang, Huazhong Wang, Xueying Wu, "Study on energy separation characteristics inside the vortex tube at high operating Pressure",TSEP Thermal Science and Engineering Progress (2019)
3) Kiran Dattatraya Devade1, Ashok T. Pise2 & Atul R. Urade3, "Numerical Analysis of Flow Behavior in Vortex Tube for Different Gases" Mechanical Engineering Research; Vol. 7 No. 2; Published by Canadian Center of Science and Education
4) 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 (2006)
5) Konstantin. Matveev , Jacob Leachman, "Numerical investigation of vortex tubes with extended vortex chambers" International Journal of Refrigeration (2019)
6) A. Agholi, M Sorin "Thermodynamic performance of a CO vortex tube based on 3D CFD flow analysis" International Journal of Refrigeration 108 (2019)
7) Mehri Alizadeh, Khosro Iraji, 3D" CFD analysis of heat transfer process inside vortex chamber of Ranque-Hilsch vortex tube- optimization of bottom radius" Progress in Solar Energy and Engineering Systems Vol. 2, No. 1, March, 2018, pp. 5-10