# OpenFOAM Case-Study Project

FOSSEE, IIT Bombay

## 3D Simulation of Flow Inside a Counter Flow Vortex Tube

**Prajit Dhakal**

Undergraduate Student, Aerospace Engineering,

Pulchowk Campus, IOE, Tribhuvan University

## Synopsis

This study presents the 3D simulation of the flow inside counter flow vortex tube also known as Ranque–Hilsch Vortex Tube (RHVT), using open source environment of *OpenFOAM*. The vortex tube is a device which separates the compressed inlet gas into stream of hot and cold gases with no moving parts and complex geometry. They are mostly used as cooling devices and used in application for spot cooling. The main objective of this study was to develop and computational environment inside *OpenFOAM* for simulating flow inside the RHVT and perform different analysis. The geometry for the problem was taken from a reference study and generated using *FreeCAD* with some modificaiton. Mesh generation was done using *snappyHexMesh* utility of and two solvers were tested for the problem formulation i.e. *rhosimpleFoam* and *sonicFoam*. Due to the presence of reverse flows at inlet and outlet, *rhoSimpleFoam* was later discarded and problem was computd using *sonicFoam* solver. For the choosen geometry, with inlet velocity of $200m/s$, cold outlet pressure of $1e^5\,Pa$, hot outlet pressure of $1.15e^5\,Pa$ and cold mass fraction of around 0.73, the total and static temperature at hot outlet is found to be $299k$ and $298k$ while at cold outlet $294k$ and $261k$ respectively.Specal care was taken for the max courant number, turbulence parameter setup and mesh generation as they diverged the solution and increased the computational time significantly. The post processing was done using open source platfoam line *Paraview* and plots were generated using *MATLAB*.

# Acknowledgement

Foremost, I would like to express my sincere appreciation to **Prof. Manimaran R**, my project supervisor, for his constant guidance, and inspiring encouragement. His valuable insights on this project and beyond have been immensely instrumental to me. I sincerely acknowledge his supervision and expertise.

I sincerely appreciate the guidance and support of **Mr. Evan Fernandes** from the FOSSEE team who has been my mentor throughout this project for his support and help during the project and for the other resources.

I extend my gratitude to the **FOSSEE, IIT Bombay** for providing me opportunity to engage in this collaborative project. This experience has allowed me to apply the knowledge acquired throughout my undergraduate study. Furthermore, This has allowed me to strengthen my own ability to apply theoretical concepts to practical applications.

**Author:**
Prajit Dhakal

# Introduction

For cooling in different applications, various devices and machines are used such as heat exchanger, vortex tubes, and so on. A vortex tube is a device that converts a compressed stream of air into stream of cold and hot air. It contains no moving parts which makes it more simpler and compact in design and effective in implementation as it is easy to use. This works by separating compressed stream of air into 2 streams with different temperature which happen due to the swirling motion generated inside the device. This swirling motion inside the tube is generated due to geometry of the tube as well as the inlet and outlet configuration of the tube. The compressed air is injected tangentially into the tube and due to its circular geometry, the compressed air tend to move tangentially which creates the swirling motion. This converts the air into two stream of different temperature flow, cold flow at the core of the tube and hot flow at the boundary of the inner surface of tube [1]. There are mainly 2 types of vortex tube based on the flow that happens inside of the tube. One is parallel flow (Uni-flow) vortex tube and counter rotating vortex tube [2]. In parallel flow vortex tube, the flow of the both stream of air i.e. hot and cold stream is at one direction. It has 2 outlet in same direction for exiting the core cold flow and outer hot flow. But in case of the counter rotating vortex tube, the flow of the cold stream and hot stream is in different direction [2]. This is shown in the figure [1] below.

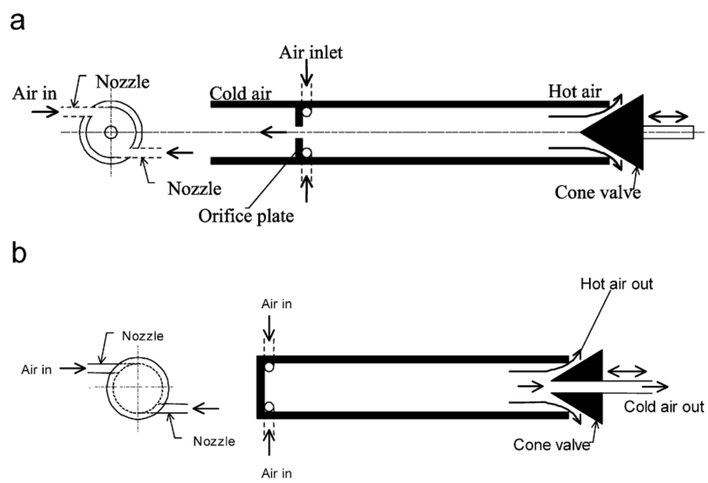



Figure 1: Types of vortex tube. a) Counter flow vortex tube and b) Parallel(Uni) flow vortex tube [3]

Despite being simple design and easy to use machine, vortex tubes are not heavily used for cooling in all application areas. This is due to its low efficiency and low coefficient of performance [4]. Many studies have been done to improve its performance and efficiency by changing different design parameters and operating conditions. This study aims simulates the 3D flow inside a counter flow vortex tube which is also called as Ranque-Hilsch Vortex Tube (RHVT) and establish environment for investigation of different studies related to vortex tube using OpenFOAM environment.

# Theoretical Background

For this study, knowledge about certain concepts are very important and needed to be taken into consideration before advancing. Some of these concepts are described below.

## Working Of Vortex Tube

The counter flow vortex tube or RHVT works is a device which separates the compressed inlet air into cold and hot stream of air at two outlets. When compressed air is injected through a inlet nozzle tangentially into the tube, it creates a high speed swirling motion inside a tube which plays the major role in separation of flow into cold and hot stream. The cold flow is at the core of the tube and hot air flow at the outer perimeter of internal surface of the tube. This tube has two outlets named as hot and cold outlet from where hot and cold air exits respectively. Hot outlet is far from the inlet and is made at so that only outer perimeter of swirling air inside the tube can exit. Cold outlet is at the end near the inlet and let the core flow exit from the tube. Due to this design, it creates temperature difference at two exit which is called as temperature separation phenomenon [5]. There is a movable nozzle at he hot exit of the tube which controls the area of hot outlet by changing its diameter. The amount of temperature difference and energy separation depends upon this area of hot outlet which affects the mass fraction of air coming out of both outlet. When compressed air is inserted form inlet nozzle tangentially, it moves towards hot outlet with high velocity and swirl. Since the area of hot outlet only let certain amount of air to flow out, remaining air will be slowed down at the hot exit and mainly at the core of the circular flow. This accumulation of air at hot outlet creates high pressure zone near the hot outlet at core region. Due to this the air begins to move in opposite direction at the core and moves towards the cold exit. Due to shear and viscosity, the outer swirling flow will induce swirl on the core flow this all phenomenon creates the energy and temperature separation inside a tube. The actual reason behind the temperature separation, which is also called as energy separation, is still unknown but there are many arguments which might explain some phenomenon going inside the tube. The tube has no moving parts and hence there in no work or energy input externally during the working of tube. Due to this it is evident that all the energy separation and other temperature difference phenomenon happens internally by some factors. Some of these arguments points that the pressure difference between the core and outer flow causes the gas to expand towards core flow and hence cools during expansion. Some suggest the cooling happens due to the contribution of shocks and acoustic phenomenon inside the tube. But the most promising explanation is given by the arguments made based on centrifugal and viscous effect inside the tube. The centrifugal theory states that during circular motion, the air particles with higher energy will mode outward due to high centrifugal force that it experience and the particles with lower energy will remain at the inner part creating high energy outer vortex and low energy core vortex. The viscous theory states that, the viscosity of the air will create the viscous dissipation between adjacent layer of the flow due to which the energy of the outer flow will rise and core flow will be less. Though experimentally none can be validated due to inability of accurate measurement of internal pressure, velocity and temperature distribution of the tube, but these phenomenon are needed for the simulation of the vortex tube [5].

## Cold Mass fraction

Mass Fraction (MF) is a dimensionless number which is used in the performance analysis of the vortex tube. It is given as the ratio of total outlet mass flow rate by total inlet mass flow rate. It is denoted by ($w$). For the vortex tube analysis, the cold mass fraction(CMF) is important which is the ratio of total mass flow rate at the cold outlet to total mass flow rate at inlet. This is denoted by symbol ($w$) and given as,

$$w = \frac{\dot{m}_c}{\dot{m}_i} \tag{1}$$

where:

$w$ Cold mass fraction (dimensionless)

$\dot{m}_c$ Mass flow rate at the cold outlet (kg/s)

$\dot{m}_i$ Mass flow rate at the inlet (kg/s)

This is an important parameter for the performance of vortex tube as this influence the amount of energy and temperature separation inside a vortex tube. This is generally controlled by changing the movable nozzle at he hot exit of tube. With optimal CMF, we can get maximum energy separation inside a tube [4].

## Turbulence Modeling

The flow inside the tube is at high speed and there is high swirling. Due to this accurate turbulence modeling of the problem is required to accurately predict the result. Various turbulent models like k-epsilon model, k-omega-SST models and so on can be used. The parameters are calculated accordingly. But among other models k-epsilon and k-omega-SST model are found to be more accurate according previous studies [6]. The parameters are calculated using the following formulas [7]:

$$k = \frac{3}{2}(U \cdot I)^2 \tag{2}$$

where:
$k$: Initial turbulent kinetic energy,
$U$: Initial velocity magnitude, and
$I$: Initial turbulence intensity, expressed as a percentage.
For pipe flows, the turbulence intensity $I$ can be estimated using the Reynolds number $Re$ based on the pipe diameter:

$$I = 0.16 \cdot Re^{-\frac{1}{8}} \tag{3}$$

The dissipation rate $\varepsilon$ is related to $k$ and the turbulence length scale $l$ by the equation:

$$\varepsilon = c_\mu^{\frac{3}{4}} \cdot k^{\frac{3}{2}} \cdot l^{-1} \tag{4}$$

where $c_\mu$ is a model parameter, typically valued at 0.09.
The turbulence length scale $l$ can be estimated as:

$$l = 0.07 \cdot L \tag{5}$$

where $L$ is a characteristic length, such as the inlet duct or pipe diameter.

## Courant Number

The Courant number, also used as the CFL number (Courant–Friedrichs–Lewy number), is a dimensionless number that is the ratio of distance travelled by some particle to the spatial grid size at one time step [8]. This is denoted by $Co$ and is given as:

$$\mathrm{Co} = \frac{u \cdot \Delta t}{\Delta x} \tag{6}$$

where:
$u$: Velocity of the flow (m/s)
$\Delta t$: Time step (s)
$\Delta x$: Spatial grid size (m)

This is important for the convergence and accuracy of time-dependent simulations. If this number is too large (generally greater than 1), information can travel across more than one cell in a single time step, leading to numerical instability and inaccurate results. So for stability and accuracy it must generally satisfy:

$$\mathrm{Co} \leq 1$$

This ensures that the simulation is stable and accurately calculates the propagation of information in the computational domain.

# Problem Statement

This study was focused to develop and investigate the flow inside a counter rotating vortex tube also known as RHVT and study some of its performance parameters. The parameter that is focused is pressure distribution, velocity distribution and temperature distribution. For this study the computational domain was taken from this [1] referenced study where the study done in *Ansys* software. Though there are different parameters to study, but the main focus will be the temperature of cold and hot outlet as it is necessary to show the energy or temperature eseperation phenomenon. Some of the geometry was dimension were changed from the original study to reduce cell size and the used computational domain is given below:

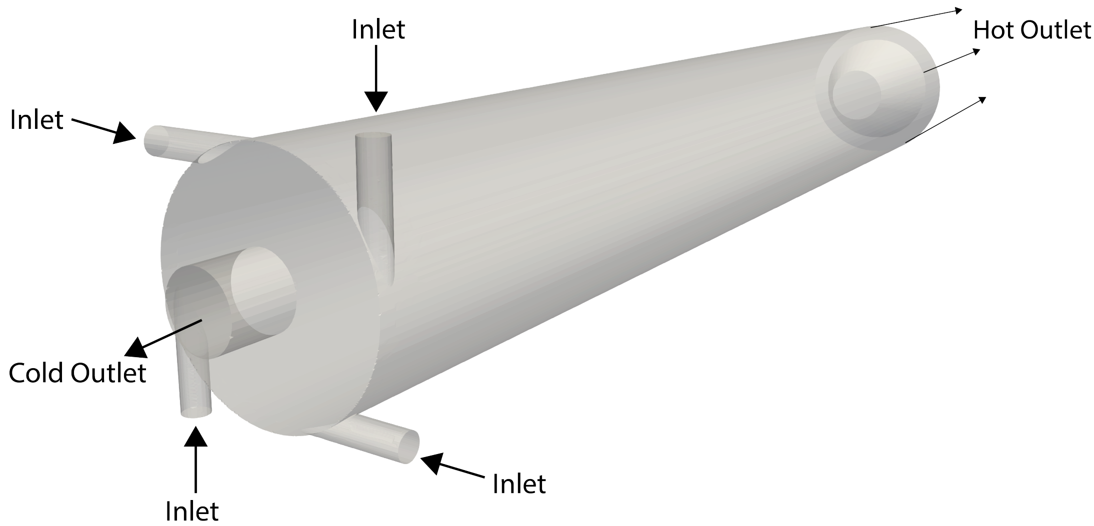



Figure 2: Computational Domain

# Governing Equations

The governing equation for the problem depends upon the solver and wat kind of simulation is being done. For this project most of the study was done using $SonicFoam$ solver for transient turbulent viscous compressible flow and hence the following are the equations that governs the calculations:

## Continuity Equation (Mass Conservation)

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0 \tag{7}$$

where:

$\rho$ : is the fluid density ($\mathrm{kg/m^3}$)

$\mathbf{u}$ : is the velocity vector (m/s)

## Momentum Equation

$$\frac{\partial (\rho \mathbf{u})}{\partial t} + \nabla \cdot (\rho \mathbf{u} \otimes \mathbf{u}) = -\nabla p + \nabla \cdot \boldsymbol{\tau} + \rho \mathbf{g} \tag{8}$$

where:

$p$ is the static pressure (Pa)

$\boldsymbol{\tau}$ is the viscous stress tensor

$\mathbf{g}$ is the gravitational acceleration vector ($\mathrm{m/s^2}$)

## Energy Equation

$$\frac{\partial (\rho e)}{\partial t} + \nabla \cdot (\rho e \mathbf{u}) = -\nabla \cdot \mathbf{q} + \boldsymbol{\tau} : \nabla \mathbf{u} - p \nabla \cdot \mathbf{u} \tag{9}$$

where:

$e$ is the specific internal energy (J/kg)

$\mathbf{q} = -k \nabla T$ is the heat flux vector

$k$ is the thermal conductivity ($\mathrm{W/(m \cdot K)}$)

$T$ is the temperature (K)

## Equation of State (Ideal Gas Law)

$$p = \rho R T \tag{10}$$

where:

$R$ is the specific gas constant ($\mathrm{J/(kg \cdot K)}$)

# Simulation Procedure

For this simulation certain process were followed to from geometry creation, mesh generation and solver setting. The initial setup file was taken form the *nacaAirfoil* simulation case already available in the *sonicfoam* solver in the tutorial of *OpenFOAM* package. The process followed for this study setup is discussed briefly below:

## Geometry

This study was mainly focused on the 3D simulation of the vortex tube. So a 3D geometry was generated using *FreeCAD* software with dimension taken form the referenced study [1]. The cold outlet length was decreased from $10mm$ to $4mm$ to decrease the mesh number. The geometry was created as a surface enclosing all sides and name was given to all the surface as shown in the figure [2] to help in the mesh generation. ll other surface that in inlet and outlet were given wall name. With that, the 3D geometry was created as shown int eh figure [3]. The dimensions of the geometry is given in the table [1] below:

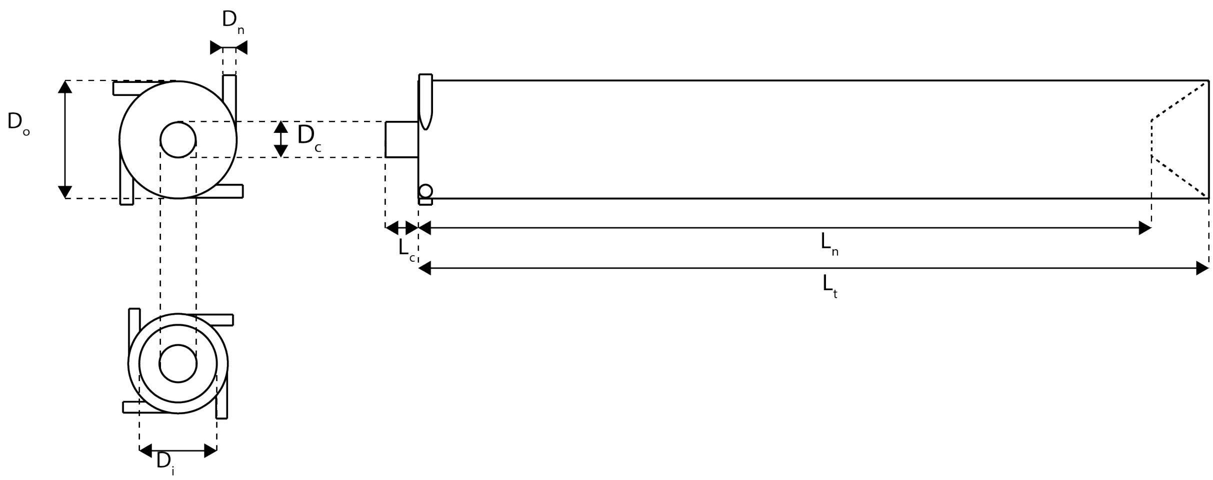



Figure 3: Geometry Details of Computational Domain

.

| **Parameter** | **Dimensions (mm)** |
|---|---|
| Inlet Nozzle Diameter ($D_n$) | 1.6 |
| Cold Outlet Diameter ($D_c$) | 5.43 |
| Tube Outer Diameter ($D_o$) | 14.48 |
| Hot Outlet Internal Diameter ($D_i$) | 11.28 |
| Cold Outlet Length ($L_c$) | 4 |
| Tube Length ($L_t$) | 97 |
| Hot Outlet Nozzle Length ($L_n$) | 90 |

Table 1: Geometric Dimensions of Vortex Tube

## Mesh

The mesh for this study was made using *SnappyHexMesh* utility available in the *OpenFOAM* package. The setup file was imported from another tutorial example with *SnappyHexMeshDict* and the geometry created was imported into a tricycle folder in the case file. Although the geometry was created with the dimensions in millimeters, during importing of the geometry into *OpenFOAM* utility, the geometry was taken as meters and all dimensions in millimeters was converted to meters. Hence *snappyHexMesh* setup was done using dimensions in meters and then later transformed into millimeters. First the surfaces were imported into the case file for meshing, then the files were merged to create the complete geometry. Also som edges were extracted which needed fine meshing for the study. Then using *snappyHexMesh*, a mesh was created as shown in the figure [4] below.

For this first a rectangular box of size $(40\,40\,140)$ was made with minimum coordinate $(-20\ -20\ -20)$ and maximum coordinate $(20\,20\,120)$. The domain was divided into $(25\,25\,60)$ grids in x, y and z direction respectively. Also, a cylindrical space was created at the core of the tube with radius of 4 to refine the core region more as the main phenomenon to be observed occurs around there. Then using *snappyHexMesh*, edge of intersection of inlet nozzle and main tube was refined to level 6. Other feature edges were refined to level 4, surfaces were refined using level $(2\,3)$ an the inside of the refinement cylinder was refined to level 2. With this, a mesh of cell number $338,456$ was created and then used for computation to capture the turbulent flow near the wall, 5 boundary layers were created with expansion ratio 1.2 and first thickness of 0.02.

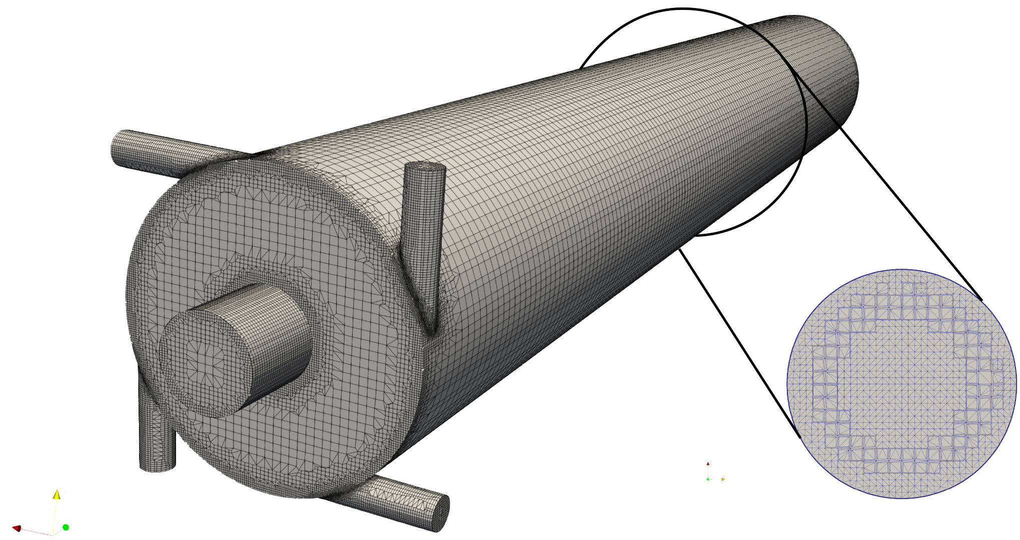


Figure 4: Mesh of Computational Domain

## Solver

The flow inside of a vortex tube is highly turbulent and rotational. Due to this accurate selection of solver si required to properly formulate the problem. For this, 2 solvers were selected among which one was *rhoSimpleFoam* and other was *sonicFoam*. *rhoSimpleFoam* is a pressure based compressible steady state solver which can solve the turbulent flow. *sonicFoam* is a pressure based compressible transient solver which can solve the turbulent flow. Due to this features and problem requirement, the problem was first solved using the *rhoSimpleFoam* solver and later only *sonicFoam* solver was used for the computation.

## Initial and Boundary Conditions

The names for the computational domain were named as shown in the figure [2]. Here, four inlet were named from one to four individually so that assigning values would be easier. All inlet, cold outlet and hot outlet were assigned as patch and remaining surface of the domain was assigned as wall and also named as wall. The initial boundary condition was assigned as given in the table below:

| | Parameter | | |
|---|---|---|---|
| **Boundary** | **p** | **T** | **U** |
| inlet_one | zeroGradient | totalTemperature uniform 300 | fixedValue uniform (0 0 0) |
| inlet_two | zeroGradient | totalTemperature uniform 300 | fixedValue uniform (0 0 0) |
| inlet_three | zeroGradient | totalTemperature uniform 300 | fixedValue uniform (0 0 0) |
| inlet_four | zeroGradient | totalTemperature uniform 300 | fixedValue uniform (0 0 0) |
| cold_outlet | waveTransmissive uniform 1e5 | inletOutlet uniform 288 | inletOutlet uniform (0 0 0) |
| hot_outlet | waveTransmissive uniform 1.15e5 | inletOutlet uniform 288 | inletOutlet uniform (0 0 0) |
| wall | zeroGradient | zeroGradient | noSlip |

Table 2: Initial boundary conditions for main field parameters

| | Parameter | | | |
|---|---|---|---|---|
| **Boundary** | **k** | $\epsilon$ | **nut** | **alphat** |
| inlet_one | fixedValue uniform 7.26 | fixedValue uniform 1737 | calculated uniform 0 | calculated uniform 0 |
| inlet_two | fixedValue uniform 7.26 | fixedValue uniform 1737 | calculated uniform 0 | calculated uniform 0 |
| inlet_three | fixedValue uniform 7.26 | fixedValue uniform 1737 | calculated uniform 0 | calculated uniform 0 |
| inlet_four | fixedValue uniform 7.26 | fixedValue uniform 1737 | calculated uniform 0 | calculated uniform 0 |
| cold_outlet | zeroGradient | zeroGradient | calculated uniform 0 | calculated uniform 0 |
| hot_outlet | zeroGradient | zeroGradient | calculated uniform 0 | calculated uniform 0 |
| wall | KqRWallFunction uniform 7.26 | epsilonWallFunction uniform 1737 | nutKWallFunction uniform 0 | compressible::alphatWallFunction uniform 0 |

Table 3: Boundary conditions for turbulence-related parameters

# Results and Discussion

The main aim of this project was to develop a 3D flow inside a counter flow vortex tube (RHVT) in *OpenFOAM* environment for further study of its field parameters and performance character. Due to this a simulation was run in 3D computational domain using *OpenFOAM*. The results were post-processed using open-source tools like *Paraview* and plots were extracted using tools like *MATLAB*.

## Solver Selection

From the referenced study [1], at the start of the simulation, the *rhoSimpleFoam* solver was used to simulate the problem. It is the pressure based steady state solver for compressible turbulent flow. But as the problem was set up in this solver, it suffered from a severe reverse flow at inlet and both outlet. The problem was first setup using pressure boundary condition at both inlet and outlet, and the problem require high pressure at hot outlet. But setting up this condition will reverse the flow from the hot outlet and the air flows inside from hot outlet. Similarly, using other boundary conditions will result in same condition and this condition persists for every boundary condition or schemes and solution methods. This might be the issue with the solver and the nature of problem to be formulated. The flow inside a vortex tube is highly turbulent and rotational. Due to this some steady state solver which takes the result of calculation from previous time step to approximate the flow and give steady results would not be able to solve the problem accurately. Due to this the there is always some reverse flow at inlet and outlets. And hence for this reason, the problem was later shifted to the *sonicFoam* solver which is pressure based transient solver for compressible turbulent flows. Since the results in the transient simulation changes with time it more accurately solves the RHVT problem and didn't have any back flow.

## Courant Number Problem

From above theoretical background, we know that the courant number plays significant role for the accuracy and stability of the computation. This dimensionless value should not be grater than 1 for good simulation and for accurate simulation, the value should range between 0.3 to 0.5. Any values greater than one will produce result that is not accurate and sometimes it will diverge the computation. In this problem, the velocity inside a vortex tube is very high i.e. sometimes it is more than the speed of sound. From equation [6], we can see that when velocity increases, the value of courant number also increases. The balancing factor in this relation is the time step which then needs to be very small. Furthermore, for the accurate solution of of the problem, the computational domain must be discretized into very small cells to reduce the error accumulation with increasing computation and to properly capture the high speed flows. For this case, the velocity of inside is also very high, and also the mesh was discretized into small cells for more accurate computation. Though it is a coarse mesh with, the value of velocity and grid size significantly increased the courant number of the flow inside the tube. Due to this, the time step that is needed for accurately solving the simulation became very small even for the coarse mesh which is in the order of $4e^{-9}$. Due to this computational time significantly increased and for the proper visualization of flow inside the vortex tube the problem was to be computed upto the time $5e^{-3}$. This made problem very tedious to develop and results needs to be visualized after the time of around 5 days. Also for the whole simulation, the max courant number was maintained below 0.2 so that the solution would not diverge and can be accurately solved with not accumulation ofthe local errors.

## Turbulence Parameter Divergence

The flow inside the vortex tube is highly turbulent. Due to this proper turbulent model and accurate turbulence parameter values are needed for the proper problem formulation. Introducing turbulence int o the flow also induce local velocity increase and also indirectly cause increase in courant number fo the flow. Therefore the value of turbulence to be used should be calculated properly which also limit the courant number to the value of 0.3 to 0.5. Generally for any turbulent problem formulation, the 5% turbulence is introduced into the flow which gives the accurate and stable turbulence modeling for most of the problem. But for this vortex tube case, introducing 5% turbulence intensity will immediately diverge the solution and especially the turbulent parameter $k$ and $\epsilon$. The values for the turbulent parameters were calculated using using a referenced resource on the web which calculates the turbulence parameter based on the velocity, turbulent intensity and characteristic length [9]. The max courant number was less than 0.2 for the simulation, and it could be increased by some value by increasing the time step for the simulation by some value. But this made the turbulent parameter diverge greatly ultimately increasing the local courant number greater than 1 and crashing the computation. Due to this only 1.1% of turbulence modeling was used for the problem and simulation was maintained below the max courant number of 0.2 for whole simulation.

With these consideration the simulation was run up to $5e^{-3}$ with time step of $4e^{-9}$. The main parameter that is used to define the phenomenon in the vortex tube is the cold mass fraction. Based on the cold mass fraction, the temperature difference and different energy separation can be gained. Also, the hottest and coldest temperature at hot and cold outlet are not gained at one cold mass fraction and it is gained at different cold mass fraction [1]. So, according to the use one optimal value of cold mass fraction is taken which given certain energy separation and temperature values at 2 outlets. Initially the cold mass fraction of the tube was stable at 0.92 which was for the time up to around 0.004. But when the flow begin to move more towards hot outlet, the values for the mass flow rate changed and the solver seems to overestimate the mass flow rate at inlet and outlet giving more outward flow than the inward flow. This needs very lon time to be stable as it seems to adjust the error little at a time. Hence taking the hot outlet amss flow rate as reference, we got the cold mass fraction of around 0.73.

The results of different field parameter inside the tube is given below. For better result interpretation, different field parameters are plotted at different axial location. These axial location are denoted by $z/L$ where $z$ is any point in axial direction and $L$ denotes the length of the tube. The values of these non-dimensional quantity are 0.15, 0.3, 0.45, 0.6, 0.75, and 0.9. At these locations, the radial variation of field parameters are plotted from center of the tube to the circumference with non-dimensional radial distance $r/R$ as x axis where, $r$ is any point in radial direction and $R$ is the radius of the tube. The field parameters are also non-dimensional where they are divided by respective parameters at inlet and plotted in y axis.
The figures below also validate the above statements made about the working of the tube and some phenomenon of the energy separation and temperature difference. According the to above statements and the referenced studies, the flow inside the tube has 2 vortices i.e. forced vortex and free vortex. Forced vortex is at the perimeter of the tube in very small area which can be seen in the different field variation. This is shown by the sharp change in the field variation at radial location around $r/R = 0.92$. The vortex formed inside this location in free vortex and the one forming outside of this location is the forced vortex. This location of free and forced vortex is more outward towards the hot end of the tube and seen more inward towards the cold outlet of the tube from figures [10] and [13]. This suggest that the area of both vortices is not uniform across the tube and vary from cold to hot outlet.

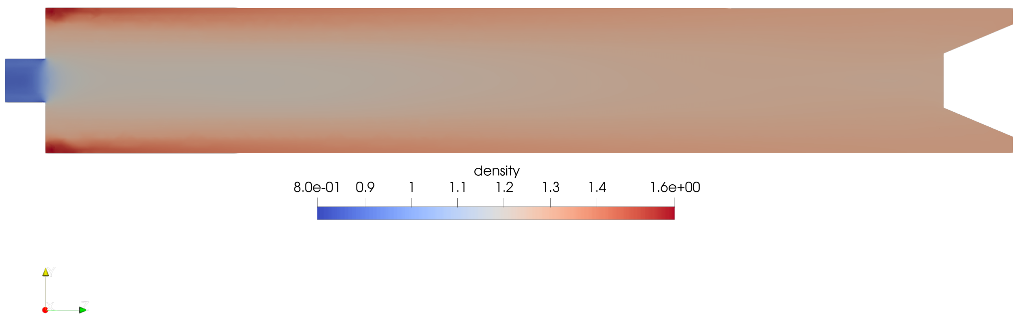



Figure 5: Density Contour at time $t = 0.005$

Figure [5] and [6] shows the density distribution inside of the tube. Here the density is higher at the outer perimeter of the tube and is less towards the center. This is due to the movement of higher energy particle towards outer perimeter and lower energy particle being at the center. This is in accordance to the statement above for the energy separation inside the vortex tube.

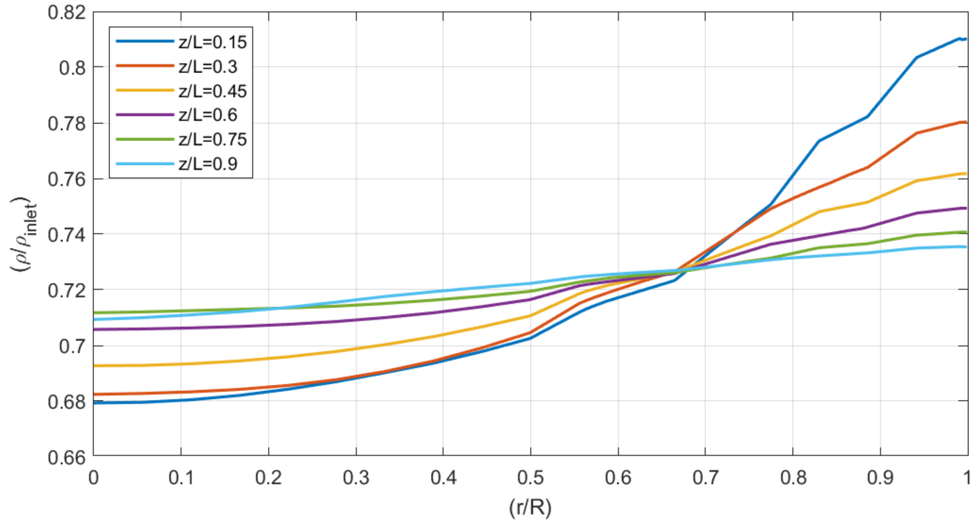
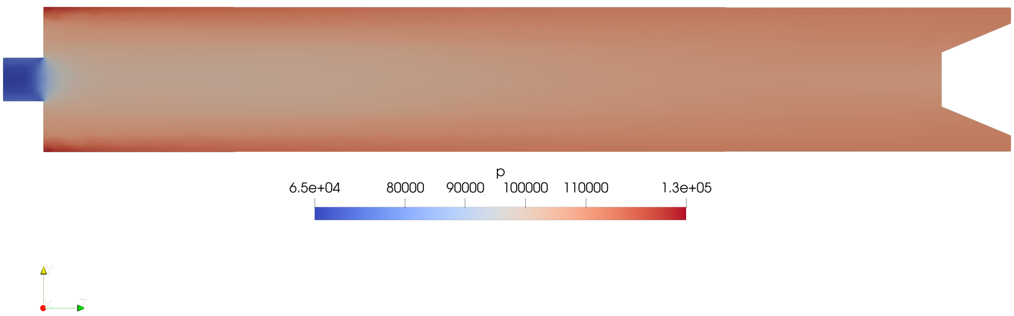
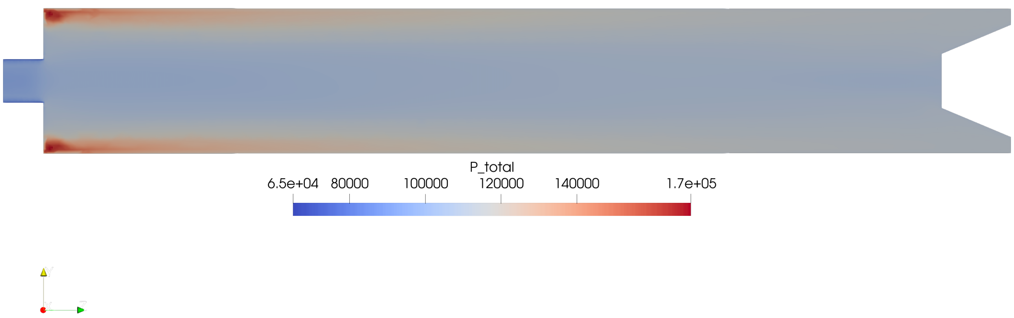



Figure 6: Radial Density Distribution at Different Points at time $t = 0.005$



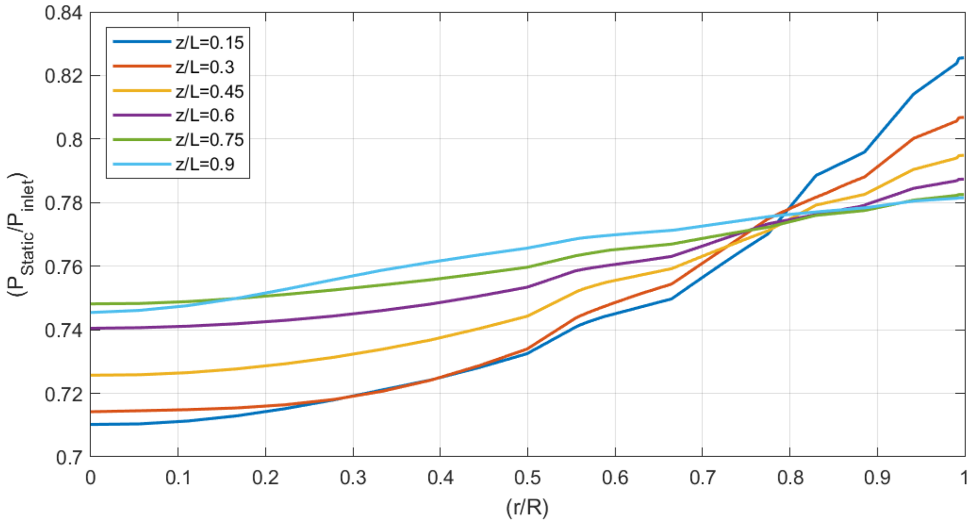

(a) Static Pressure Contour



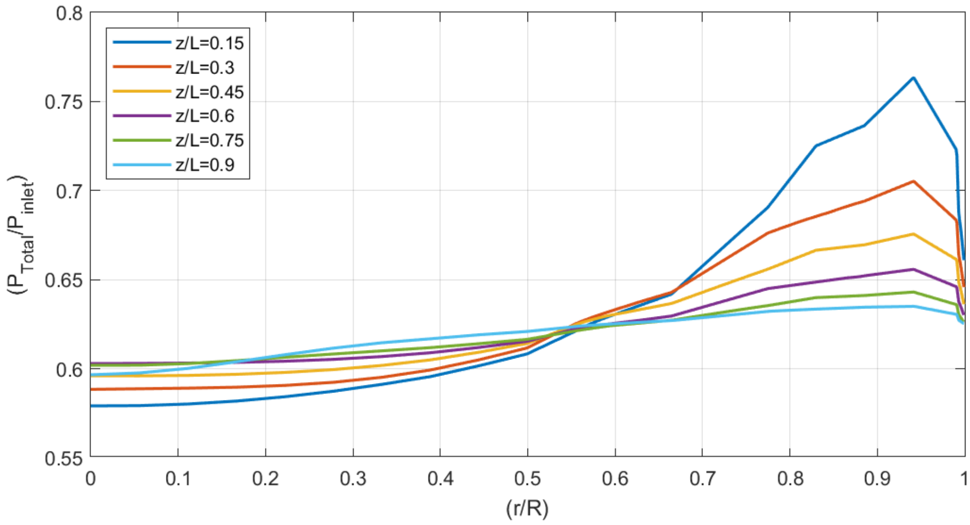

(b) Total Pressure Contour

Figure 7: Pressure Distribution at time $t = 0.005$

Figure [7] and [8] shows the pressure distribution inside a vortex tube. Here as we can see, there is low pressure towards the center of th tube which is the cor of free vortex. Due to this the highly dense particles tends to move towards the core of the tube and due to this fluid tends to expand. This creates high temperature region towards the perimeter of the tube and low temperature region towards the core of the tube due to expansion phenomenon. Also, we can see form the pressure results that the pressure has higher gradient towards the cold end of the tube which is seen as sharp rise in static and total pressure at axial location near the cold end. Towards the hot end, the velocity and density of the fluid keeps decreasing and this leads to more gradual distribution of the pressure in radial direction. Also the pressure in the core region at towards the hot outlet is much higher than at the cold outlet. This will force the air to move towards the cold outlet creating counter rotating phenomenon and hence creating the whole vortex energy separation phenomenon.

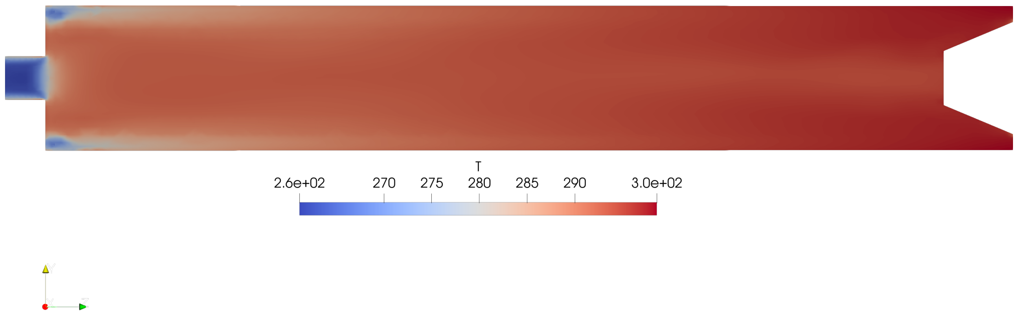



(a) Static Pressure Plot

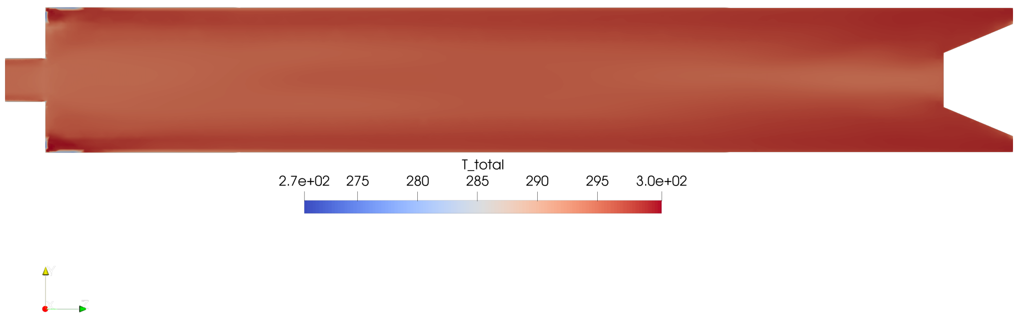



(b) Total Pressure Plot

Figure 8: Radial Pressure Distribution at Different Points at time $t = 0.005$



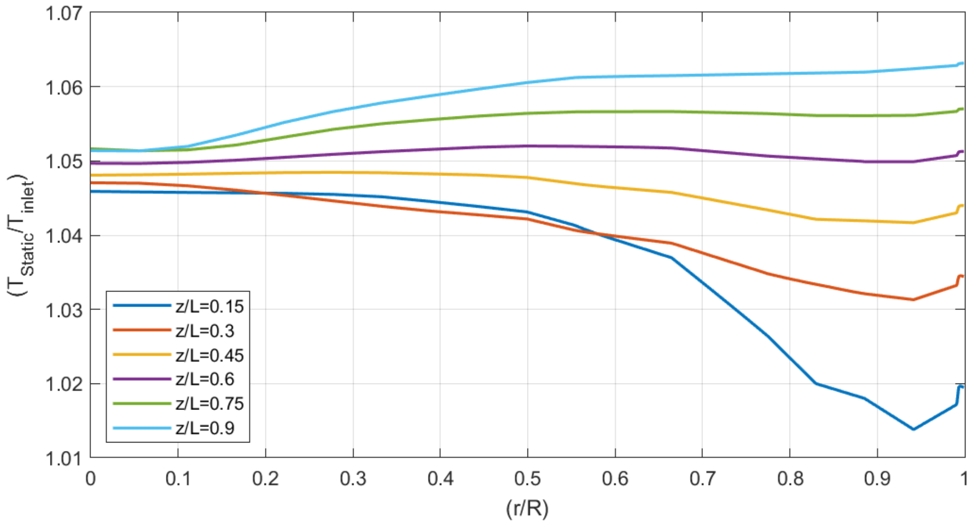

(a) Static Temperature Contour



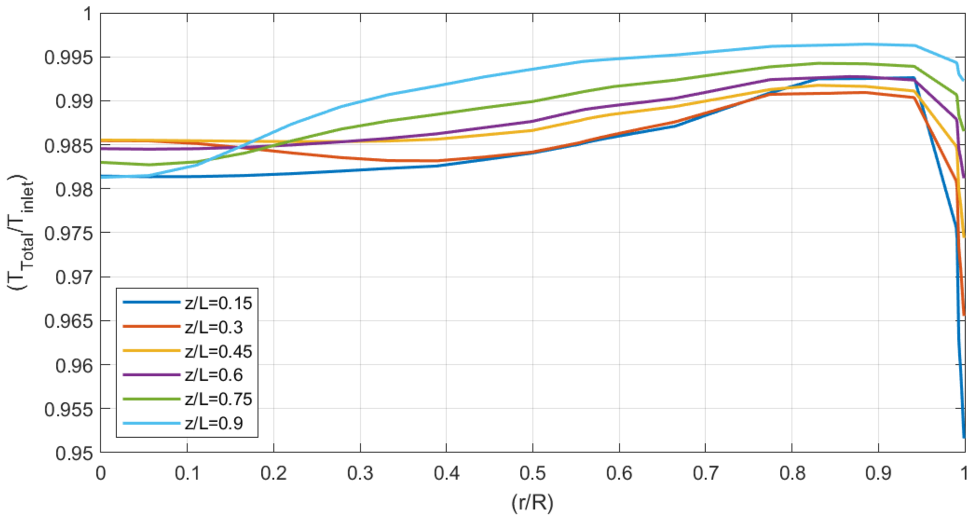

(b) Total Temperature Contour

Figure 9: Temperature Distribution at time $t = 0.005$

Figure [9] and [10] shows the temperature variation inside the RHVT. At the inlet, due to the expansion of fluid, the static pressure drops near cold outlet and it increases as the fluid moves towards the hot outlet. But the total temperature doesn't suffer much loss as compared to the static temperature. This start off at value near the inlet value and later on keeps increasing as fluid moves towards the hot outlet. This is due to the exchange of energy from the core vortex moving towards cold outlet to the outer vortex moving towards hot outlet. This is due to the viscosity of the fluid and highly turbulent nature of fluid moving inside the tube. This can also be seen in figure [11] where different turbulent parameters distributions are shown. Here, the turbulent kinetic energy contour, which shows the intensity of turbulence inside the tube, shows that there is high turbulence near the cold outlet region and also towards the middle of the tube axially. Due to this there is higher exchange and dissipation of energy among the fluid particle near that region due to turbulence. This is also supported by the turbulent viscosity contour and turbulent thermal diffusion contour below. Here, we can see that the due to turbulence the viscosity of the fluid at the core region near the cold outlet increases. This happens due to change in momentum due to turbulence. This creates more rigorous interaction of fluid particle between core flow and outer vortex. Due to this there is more energy exchange and diffusion. Also, the turbulent thermal diffusivity shows that there is higher thermal dissipation at the core vortex region near the cold outlet. Due to all this the energy of the core region is transferred to the outer vortex making it rise in temperature as the core flow loses the temperature. From the plots of temperature below, we can see that there is steep radial distribution of temperature towards the cold end and more gradual distribution towards the hot end. This is because of the loss in tangential velocity near the hot outlet and less turbulence which contributes majorly in the temperature difference and energy separation. The total and static temperature at hot outlet is found to be 299$k$ and 298$k$ while at cold outlet 294$k$ and 261$k$ respectively.

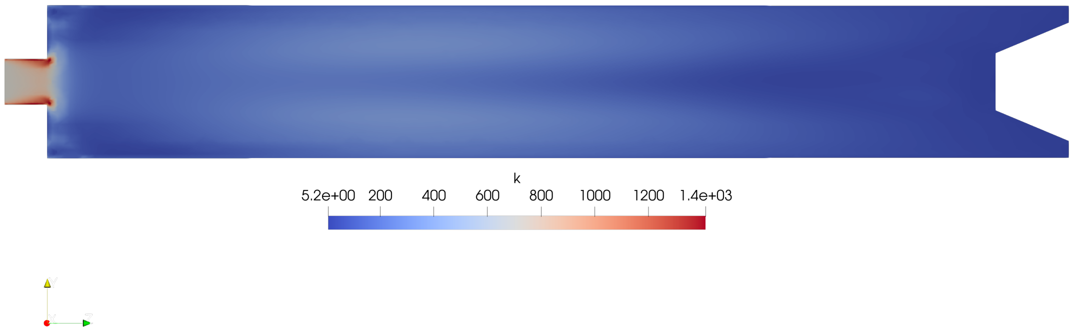



(a) Static Temperature Plot

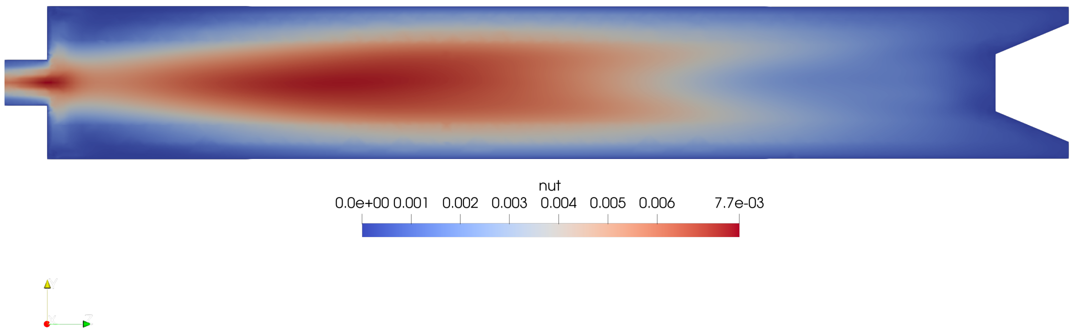



(b) Total Temperature Plot

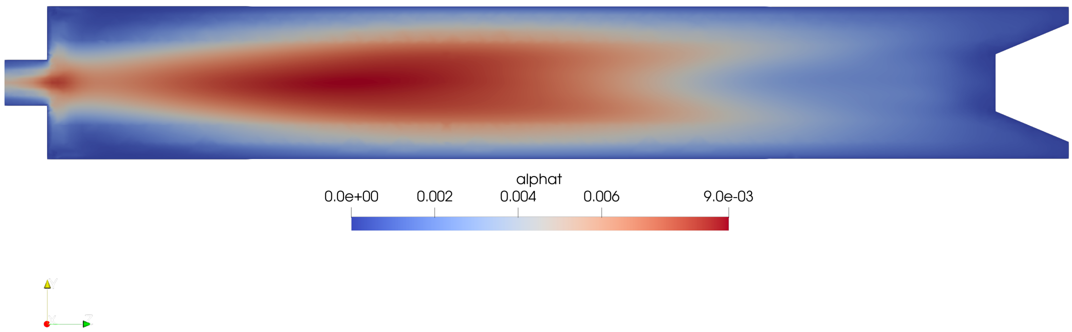
Figure 10: Radial Temperature Distribution at Different Points at time $t = 0.005$




(a) Turbulent Kinetic Energy Contour



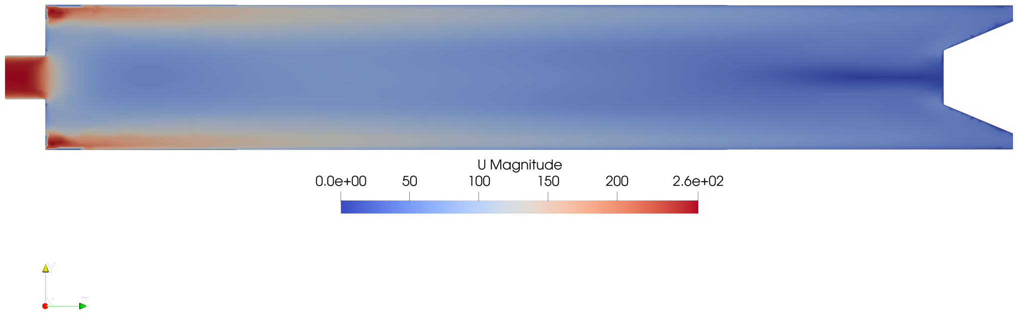

(b) Turbulent Viscosity Contour



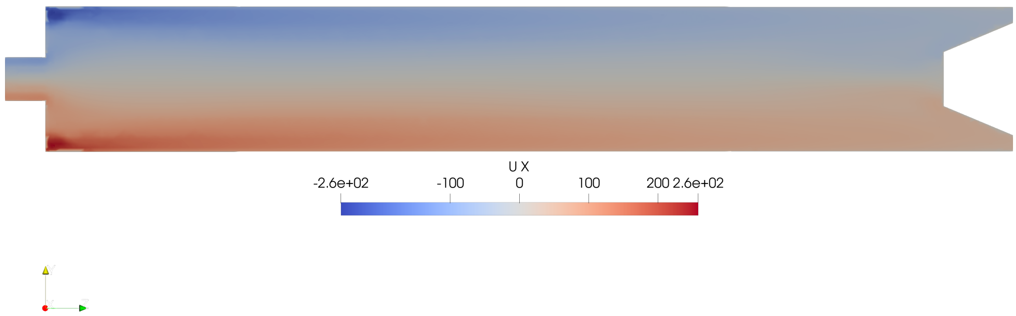

(c) Turbulent Thermal Diffusivity Contour

Figure 11: Turbulent Parameter Distribution at time $t = 0.005$

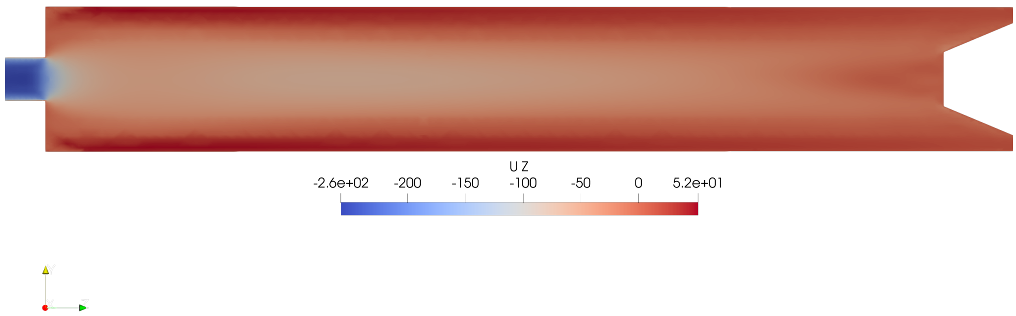
Figure [12] and [13] shows the velocity distribution inside the tube. Here we can see that the velocity inside the tube is very high and is the major contributor to energy separation inside the tube. The velocity near the inlet and at the outer vortex is high at the beginning and it decreases moving towards the hot outlet. At the core region near the hot outlet the velocity becomes zero which gives to the pressure rise and the fluid moves towards the cold outlet creating core vortex, hence generates the counter rotating vortices. This also creates more steep variation of radial velocity near cold outlet and more gradual increase towards the hot outlet as seen in the plots below.




(a) Velocity Magnitude Contour



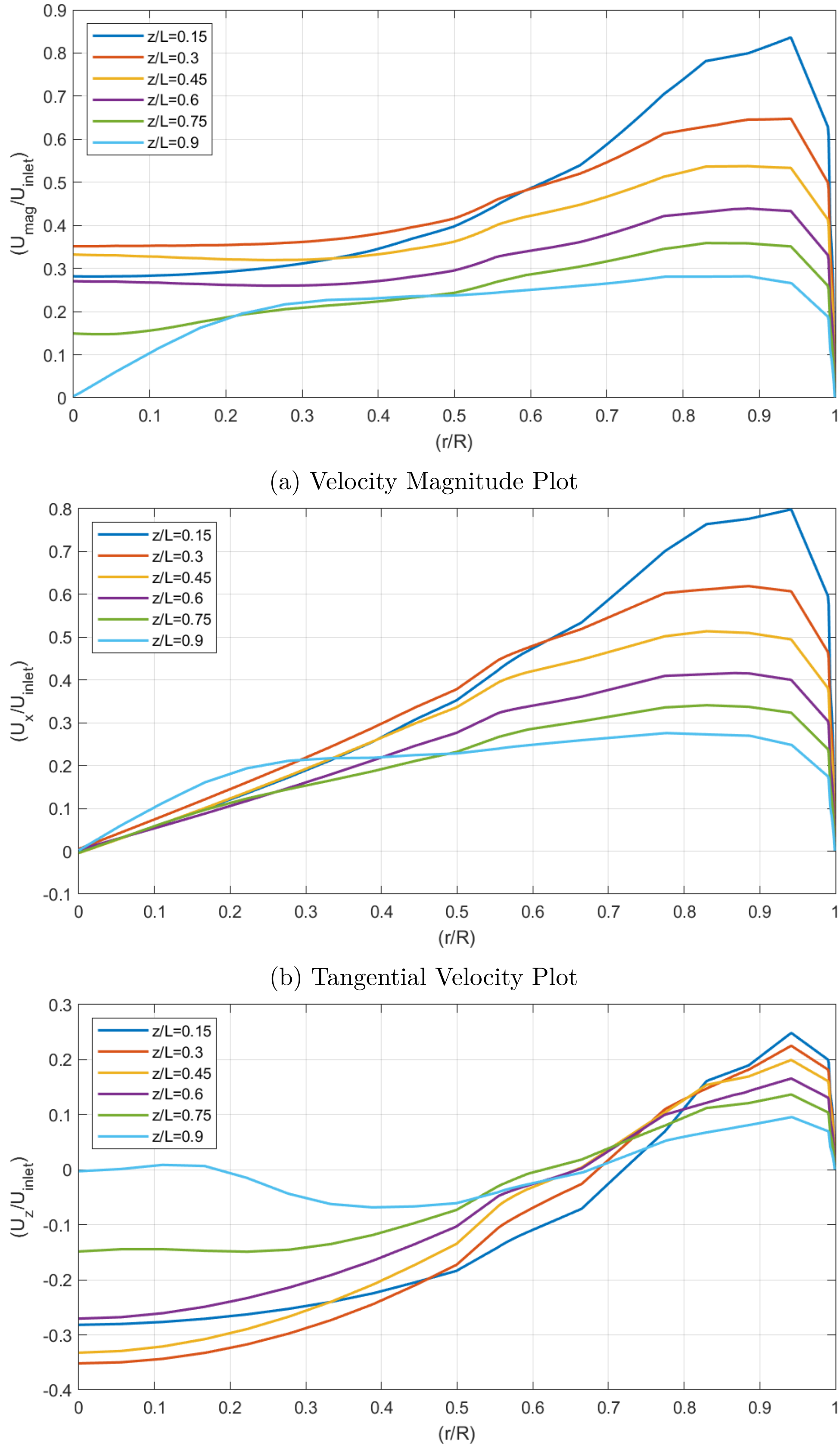

(b) Tangential Velocity Contour




(c) Axial Velocity Contour

Figure 12: Velocity Distribution at time $t = 0.005$

From the tangential and axial velocity plots given below, we can see that tangential component of velocity is much higher than that of the axial component. This shows that the major flow happens due to the tangential velocity inside the tube. Also, from the plots below and contours above, we can see that the tangential component of velocity is very high towards the outer perimeter of tube and it dies out moving towards the center. Due to this, the major flow of outer vortex is governed by the tangential component of velocity and the core flow is majorly governed by axial component of velocity. The streamlines and formed vortices are shown in the figure [14] below.




(a) Velocity Magnitude Plot

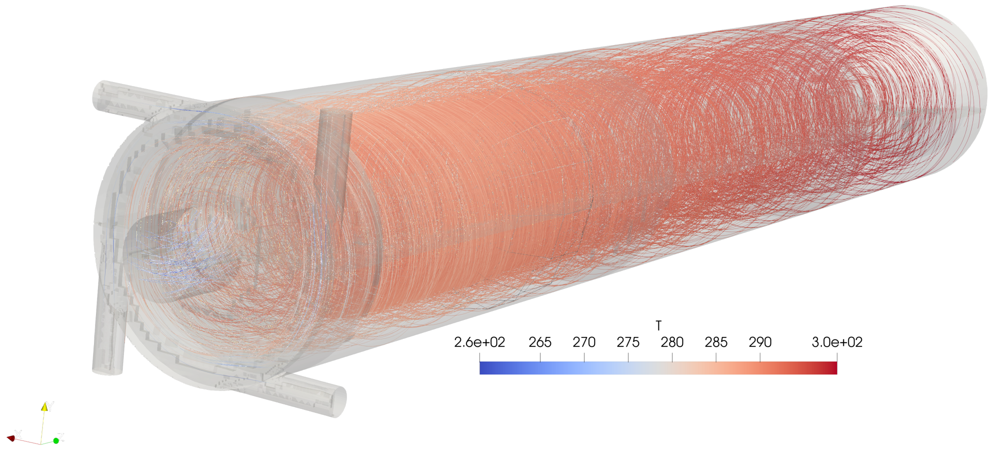


(b) Tangential Velocity Plot

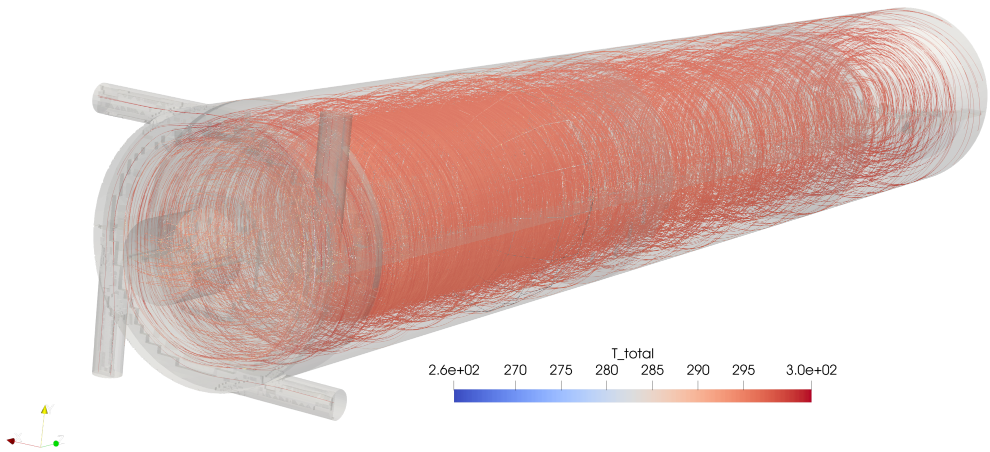


(c) Axial Velocity Plot

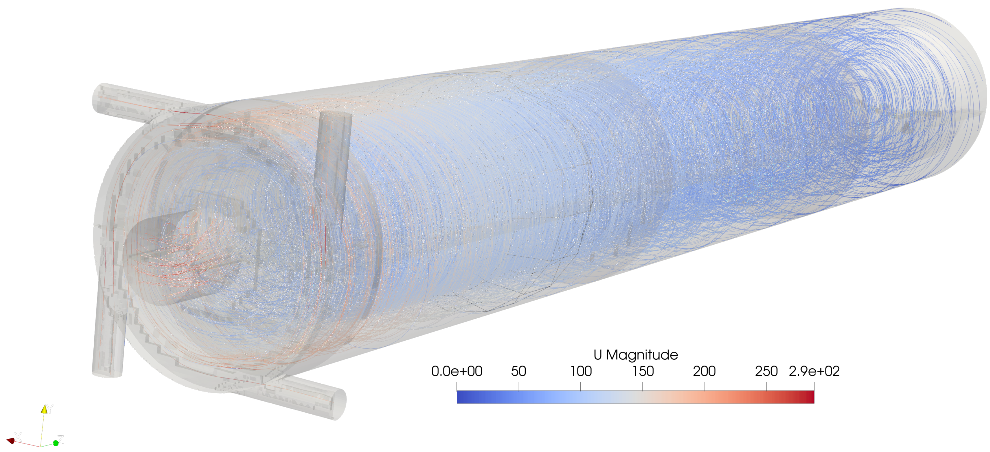
Figure 13: Radial Velocity Distribution at Different Points at time $t = 0.005$




(a) Streamline of Static Temperature




(b) Streamline of Total Temperature




(c) Streamline of Velocity Magnitude

Figure 14: Turbulent Parameter Distribution at time $t = 0.005$

.

# Validation and Comparison

All the above results were calculated form the coarse mesh. This wa because of the computational time that the problem took. For the cell number of $338,456$, it took around 5 days to complete the simulation to time 0.005. Also to make the solution not to diverge and being stable and accurate courant number was maintained below 0.2. Due to all these restraints, the time step became very small even for the coarse mesh and took significantly long time. Due to this grid independence test was not carried out as more fine mesh would require much small time step to maintain the courant number. Also, the referenced study [1] from which geometry was taken didn't operated in the similar initial parameter as this study. It operated in much higher initial condition then this study parameter. It was not possible to specify that condition in this study because of high computational time it needed and the problem of solution and turbulent parameter diverging in this environment. But th nature of field variation inside the tube is matching which validates the flow. Results might not be very accurate as the results are deducted from the coarse mesh but the nature of the flow aligns with the reference study [1]. Also, there is a study [6] which is using nearly the same operating condition as this study but the problem formulation is in 2D environment and also have very high length to diameter ratio. Though it cannot be used accurately fo validation, the value of hot and cold outlet temperature obtained this referenced study is very close to the value of this study. Due to this the study was validated and was considered satisfying for this study. Although with higher computational power and proper mesh domain, this can be validated using grid independence tests and other validation.

# Conclusion

In this project, 3D simulation of the flow inside counter rotating vortex tube was developed successfully enabling analysis of different flow and field parameters. Different field parameters were analyzed and compared to describe the mechanism of energy separation inside the tube. The results were interpreted in the form of different contours and plots. From this study, some of the main concluding factor that affect the problem formulation listed below:

- The use of *rhoSimpleFoam* which is pressure based steady state solver for compressible turbulent flow made is not recommended as it suffers from reverse flow due to complex rotational and highly turbulent flow with high speed inside the tube.
- The turbulent intensity should be kept very low and gradually increased to ensure that the problem doesn't diverge for the operating condition.
- Since the flow is at high velocity, even for the coarse mesh, the computational time is very high while limiting the max courant number under 1 and also using the transient solver i.e. *sonicFoam*.
- The balance of mass conservation takes long time even if the converge value are met during calculation which make simulation more time consuming.
- The operating initial value and turbulent values should be selected carefully to give minimum computational time with results that can be validated and compared.

The analysis from other study [4] shows that the performance and energy separation can be made better in the tube by changing inlet pressure, inlet velocity, hot outlet pressure, length to diameter ratio, number of inlet nozzle and so on. This study only aimed to develop the flow inside RHVT in *OpenFOAM* to provide environment for further study and with this completion the project was concluded.

# References


[1] A. AlSaghir, M. Hamdan, and M. Orhan, "Evaluating velocity and temperature fields for ranque-hilsch vortex tube using numerical simulation," *International Journal of Thermofluids*, vol. 10, p. 100074, 03 2021.

[2] E. dos Santos, C. Hood Marques, G. Stanescu, L. Isoldi, and L. Rocha, *Constructal Design of Vortex Tubes*, pp. 259 – 273. 03 2013.

[3] S. Eiamsa-ard and P. Promvonge, "Review of ranque–hilsch effects in vortex tubes," *Renewable and Sustainable Energy Reviews*, vol. 12, no. 7, pp. 1822–1842, 2008.

[4] M. Sungur, E. Elnajjar, M. O. Hamdan, and S. A. Al-Omari, "A numerical analysis investigation to optimize the performance of the ranque–hilsch vortex tube by changing the internal tapering angle," *International Journal of Thermofluids*, vol. 20, p. 100467, 2023.

[5] Y. Xue, M. Arjomandi, and R. Kelso, "A critical review of temperature separation in a vortex tube," *Experimental Thermal and Fluid Science*, vol. 34, pp. 1367–1374, 11 2010.

[6] J. Burazer, A. Cocic, and M. Lecic, "Numerical research of the compressible flow in a vortex tube using openfoam software," *Thermal Science*, vol. 20, pp. 195–195, 01 2016.

[7] Wikipedia contributors, "Turbulence kinetic energy." `https://en.wikipedia.org/wiki/Turbulence_kinetic_energy`, 2025. Last revised June 16, 2025; accessed June 21, 2025.

[8] CFD Land, "What is a courant number (cfl number)?," 2020. Accessed: 2025-06-21.

[9] Wolf Dynamics, "Calculator for the estimation of turbulence properties values." `https://www.wolfdynamics.com/tools.html?id=110`, n.d. Accessed: 2025-06-21.