Fluid-Structural Analysis of Vortex Tube

Nasser Saad Abuhaimed and Khalid Hassan Alsubaie
Supervisor: Dr. Nashmi Alrasheedi and Dr. Rashid Khan
Mechanical Engineering Department, College of Engineering
Al Imam Mohammad Ibn Saud Islamic University

ABSTRACT

This study employs computational fluid dynamics using ANSYS Fluent to analyses straight and curved vortex tubes. By coupling energy, momentum, and continuity equations, we derive velocity, pressure, and temperature profiles. Objectives encompass model validation, mesh sensitivity analysis, and exploring hot outlet pressure impacts on vortex tube parameters. Results align with reference work, confirming accuracy. Mesh independence testing identifies optimal resolution, and variations in hot outlet pressure reveal potential applications for improved heating efficiency, contributing valuable insights for thermal management and energy efficiency.

MOTIVATION

Energy harvesting using vortex tubes involves harnessing the temperature differentials created within the device to generate usable energy such as: -

- Thermal Electric Generators (TEGs) : Semiconductor TEGs transform temperature differences into efficient electrical power.
- Stirling Engine Integration : Integrates vortex tube with Stirling engine for electricity using temperature differentials.

OBJECTIVES

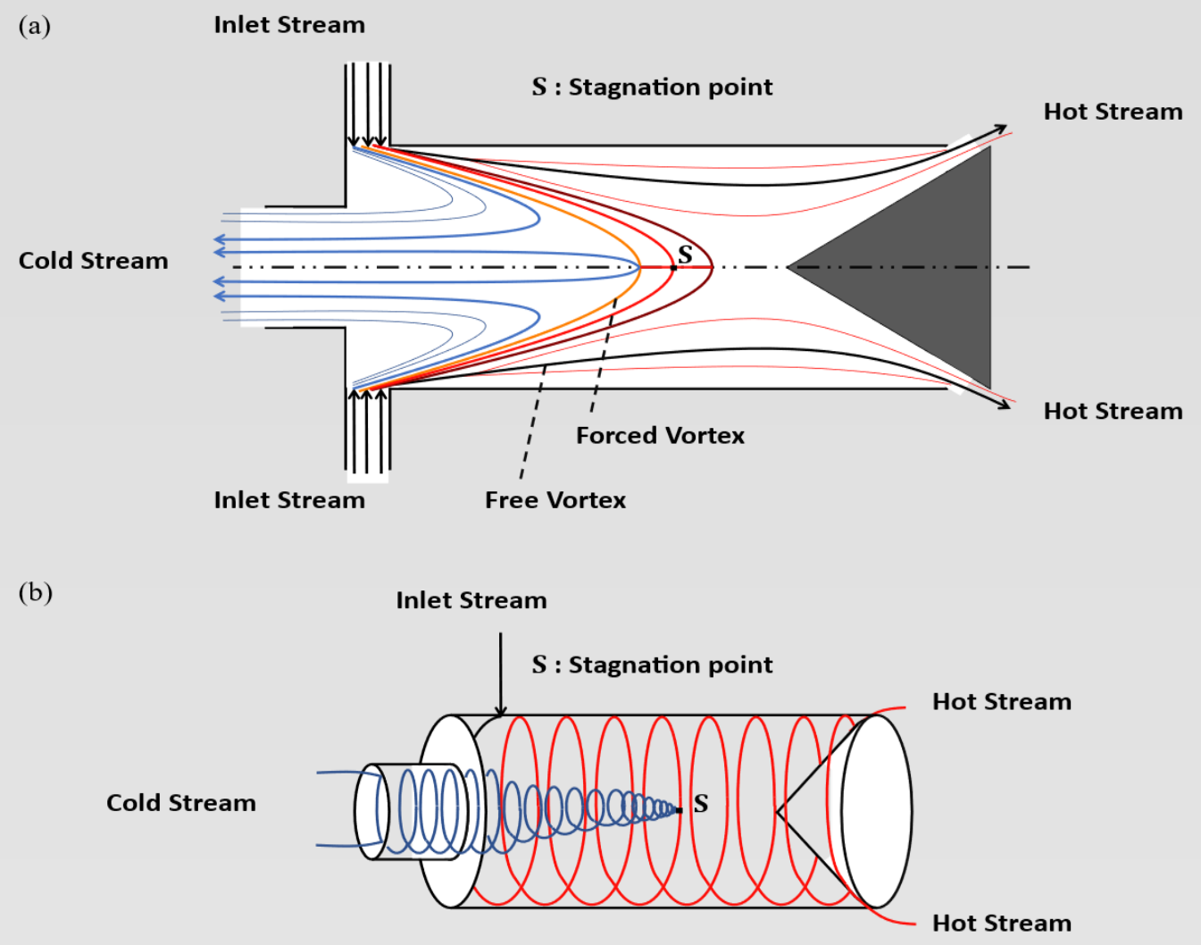
The main objectives of this work are to validate our model results, to perform the mesh sensitivity analysis, and finally to study the effect of the outlet pressure change on the main vortex tubes parameters.

METHODOLOGY

- The project's methodology employs a comprehensive computational fluid dynamics (CFD) approach using ANSYS Fluent. Analysing both straight and curved vortex tubes,
- The simulation couples energy, momentum, and continuity equations to derive vital velocity, pressure, and temperature profiles.
- Model validation involves comparing results with reference work, encompassing experimental and numerical data.
- A meticulous mesh sensitivity analysis balances accuracy and computational efficiency. A thorough literature review establishes a robust foundation, citing pertinent references. The numerical modelling phase extends existing models, and ANSYS finite element software is used for problem modelling.
- The methodology concludes with a detailed analysis of results, scrutinizing findings against reference work, contributing significantly to vortex tube performance optimization. This systematic approach covers model validation, numerical extension, finite element modelling, and results analysis.

VORTEX TUPE PROCESS

The phenomenon of energy separation takes place when a pressurized gas is injected through the inlet nozzle. This gas moves towards the hot annular outlet, forming a free vortex, while the backflow from the conical valve towards the cold outlet forms a forced vortex. The interaction between these free and forced vortices results in a stagnation point with zero local velocity, as Fulton [2] explained



MATHEMATICAL MODELING

❖NUMERICAL MODELLING

$$\frac{\partial(\rho v_i)}{\partial x_i} = 0 \qquad \text{(Continuity equation)}$$

$$\frac{\partial(\rho v_i v_j)}{\partial x_i} = \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial v_i}{\partial x_j} + \frac{\partial v_j}{\partial x_i} - \frac{2}{3}\delta_{ij}\frac{\partial v_k}{\partial x_k}\right)\right] - \frac{\partial P}{\partial x_j} + \frac{\partial}{\partial x_j}\left(-\rho\overline{v_i' v_j'}\right) \qquad \text{(Navier–Stokes equation)}$$

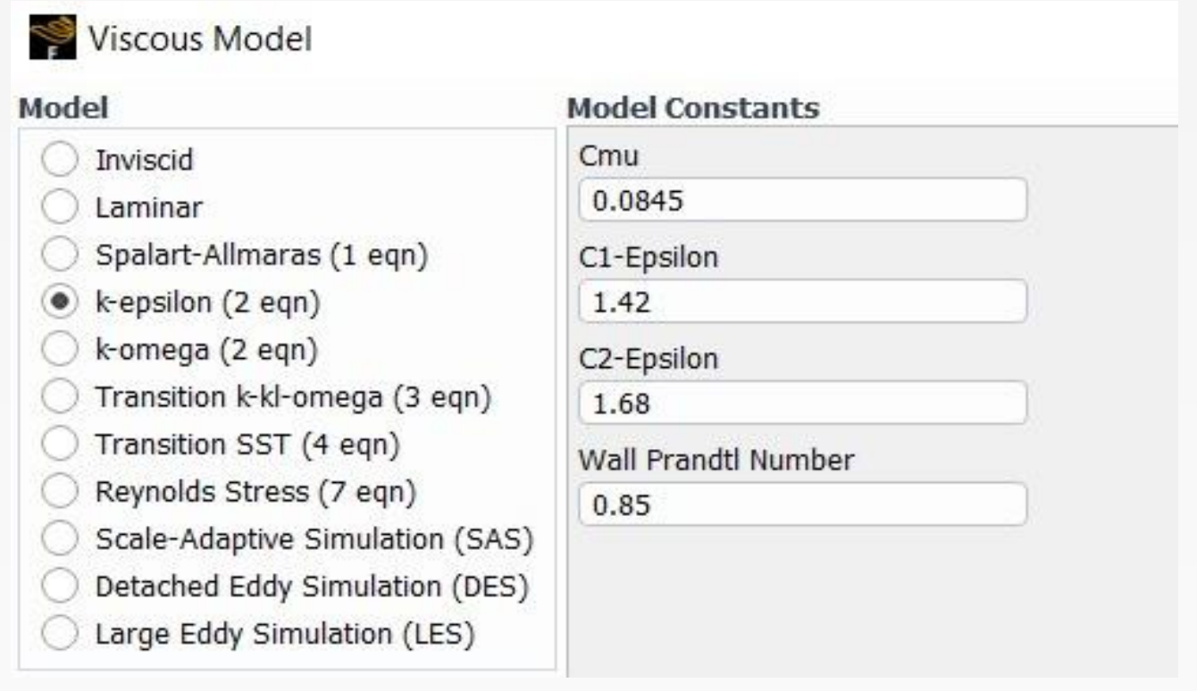
$$\frac{\partial(\rho v_i k)}{\partial x_i} = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + \mathrm{G}_k + G_b - \rho\varepsilon - Y_M \qquad \text{(Turbulence kinetic energy)}$$

$$\frac{\partial(\rho v_i \varepsilon)}{\partial x_i} = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + C_{1\varepsilon}\frac{\varepsilon}{k}(G_k + C_3 G_b) - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} \qquad \text{(Turbulence dissipation rate)}$$

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \qquad \text{(Turbulence dissipation rate)}$$

$$P = \rho R T \qquad \text{(Ideal gas law)}$$

Where:
T =Temperature [K]
P = Pressure [Pa]
c_p = Specific heat at constant pressure $[J\ kg^{-1}K^{-1}]$
K = Turbulent kinetic energy $[m^2 s^{-2}]$
V = Velocity $[m\ s^{-1}]$
P = Density $[kg\ m^{-3}]$
μ = Dynamic viscosity $[kg\ m^{-1}s^{-1}]$
ν = Kinematic viscosity $[m^2 s^{-1}]$
ε = Turbulent dissipation rate $[m^2 s^{-3}]$


Viscous Model

Model
○ Inviscid
○ Laminar
○ Spalart-Allmaras (1 eqn)
● k-epsilon (2 eqn)
○ k-omega (2 eqn)
○ Transition k-kl-omega (3 eqn)
○ Transition SST (4 eqn)
○ Reynolds Stress (7 eqn)
○ Scale-Adaptive Simulation (SAS)
○ Detached Eddy Simulation (DES)
○ Large Eddy Simulation (LES)

Model Constants
Cmu: 0.0845
C1-Epsilon: 1.42
C2-Epsilon: 1.68
Wall Prandtl Number: 0.85

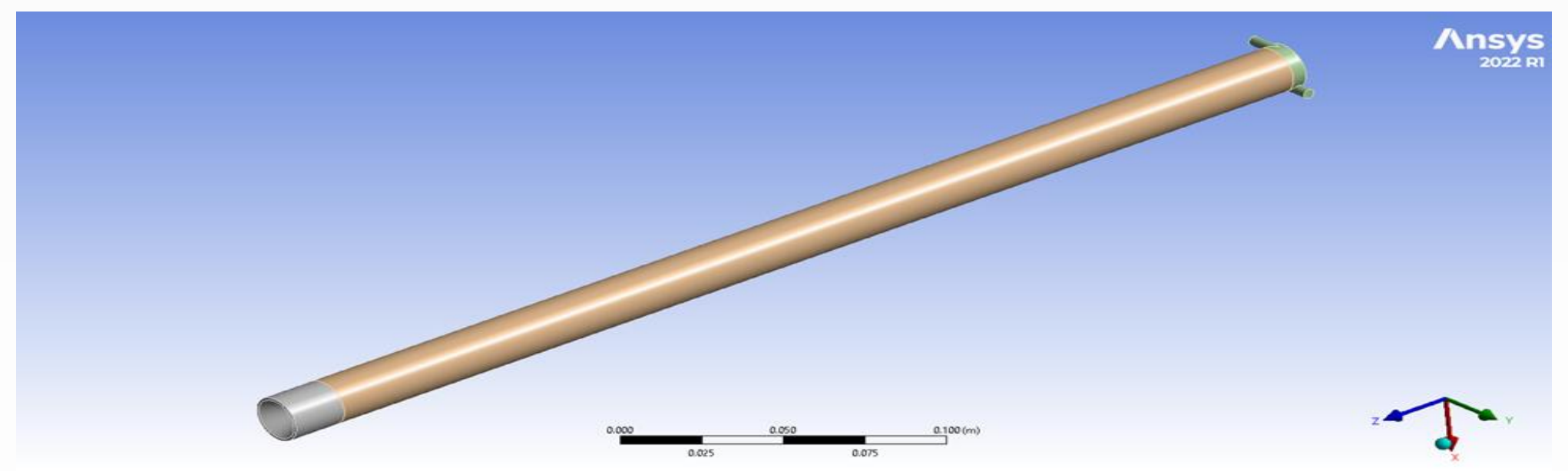
Turbulence model applied for both geometries

FINITE ELEMENT MODELING

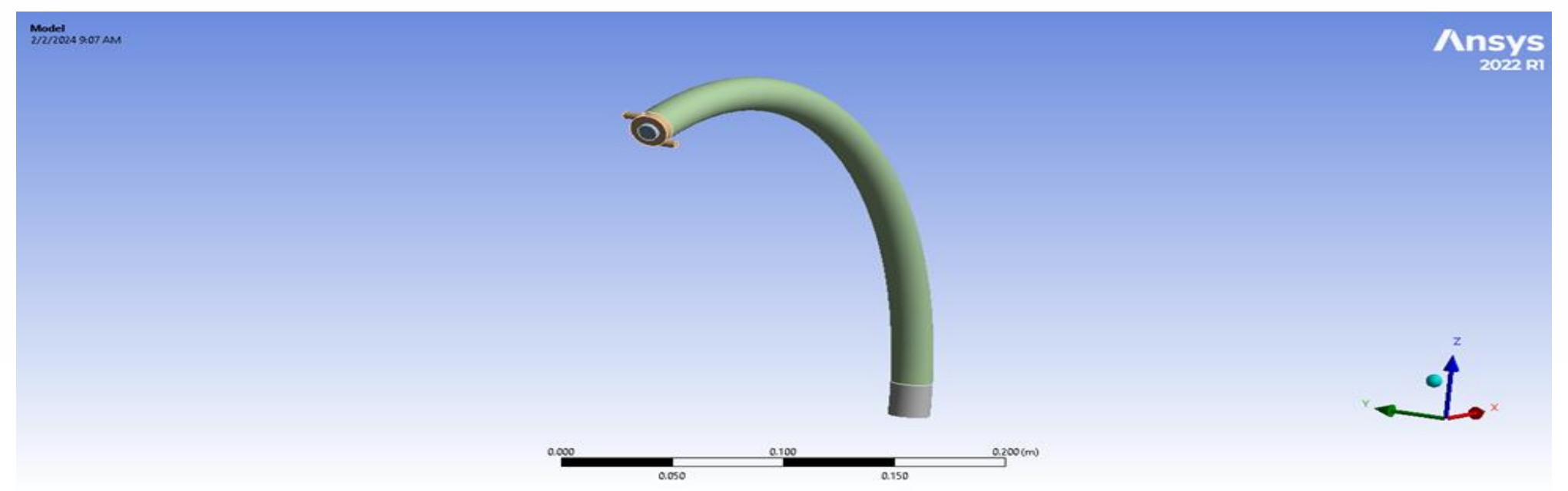
❖ Geometry

Geometry specifications for vortex tubes [1]

Geometric Parameter	Dimension
Working tube length (L)	400 mm
Inner tube diameter (D)	19.05 mm
Cold exit diameter (d_c)	9.53 mm
Nozzle diameter (d_n)	4.00 mm
Hot exit area (A_h)	58.17 mm^2
Nozzle total inlet area (A_n)	25.13 mm^2



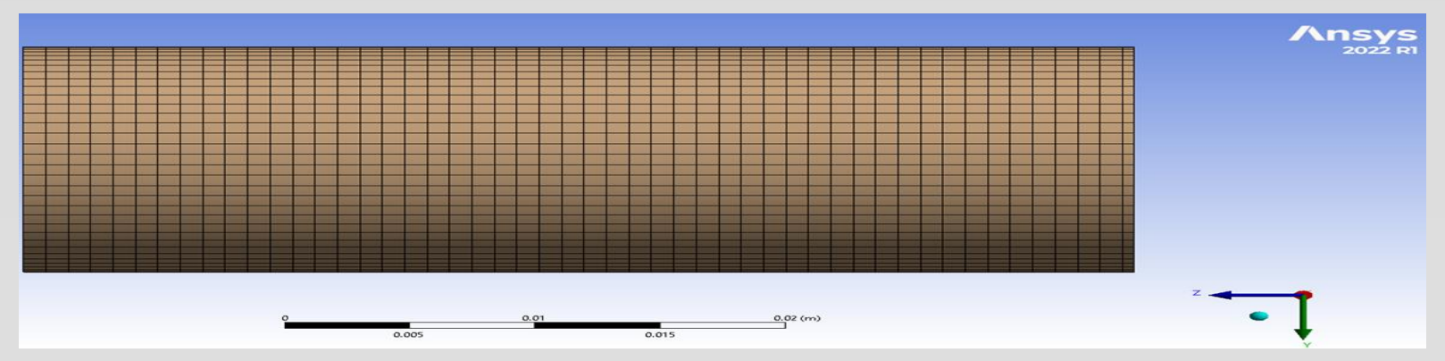
Straight vortex tube geometry



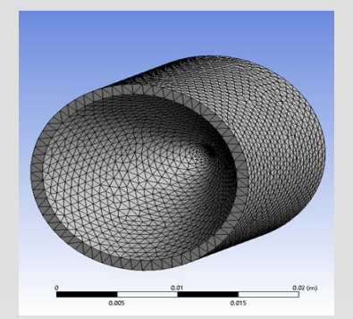
Curved vortex tube geometry

❖ Meshing

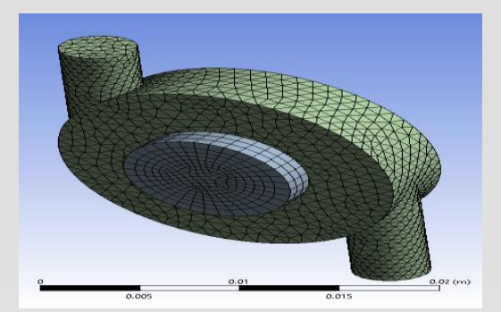
- The non-structured mesh was built to be able to reduce the total number of mesh element instead of using tetrahedral mesh for the whole geometry.
- The meshing process was performed in three steps.
- First is to apply the sweep method on the tube body
- Second was to apply inflation
- Third was to apply the tetrahedral mesh elements on the remaining



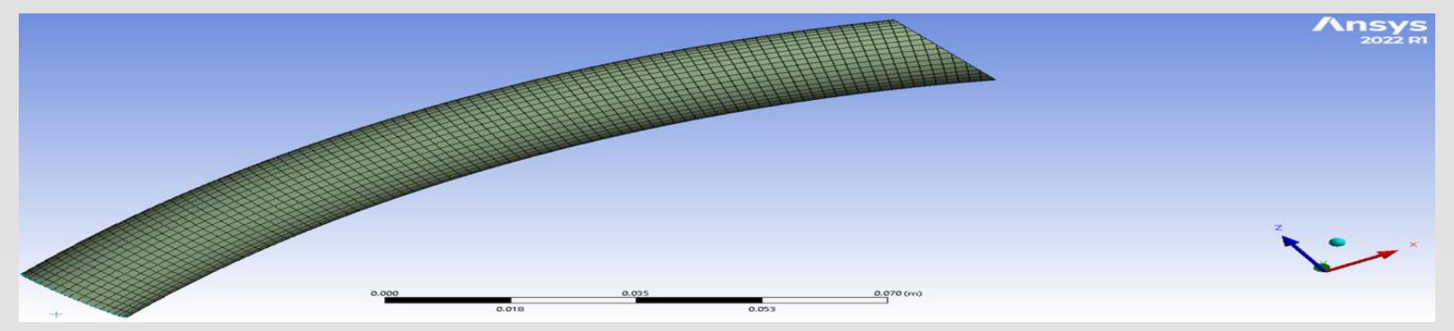
Mesh layout for straight vortex tube



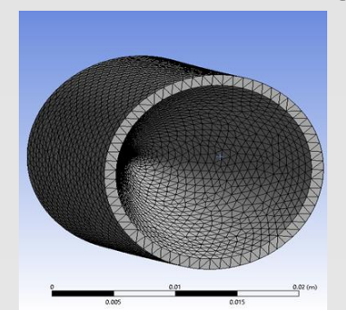
Hot outlet mesh



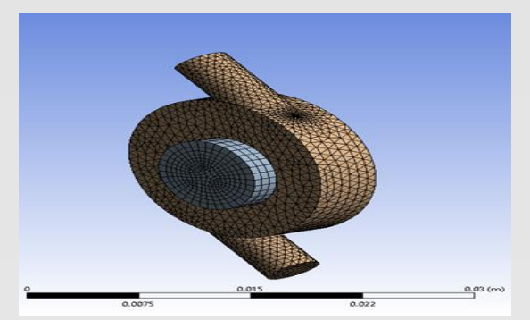
Inlet mesh



Mesh layout for curved vortex tube



Hot outlet mesh

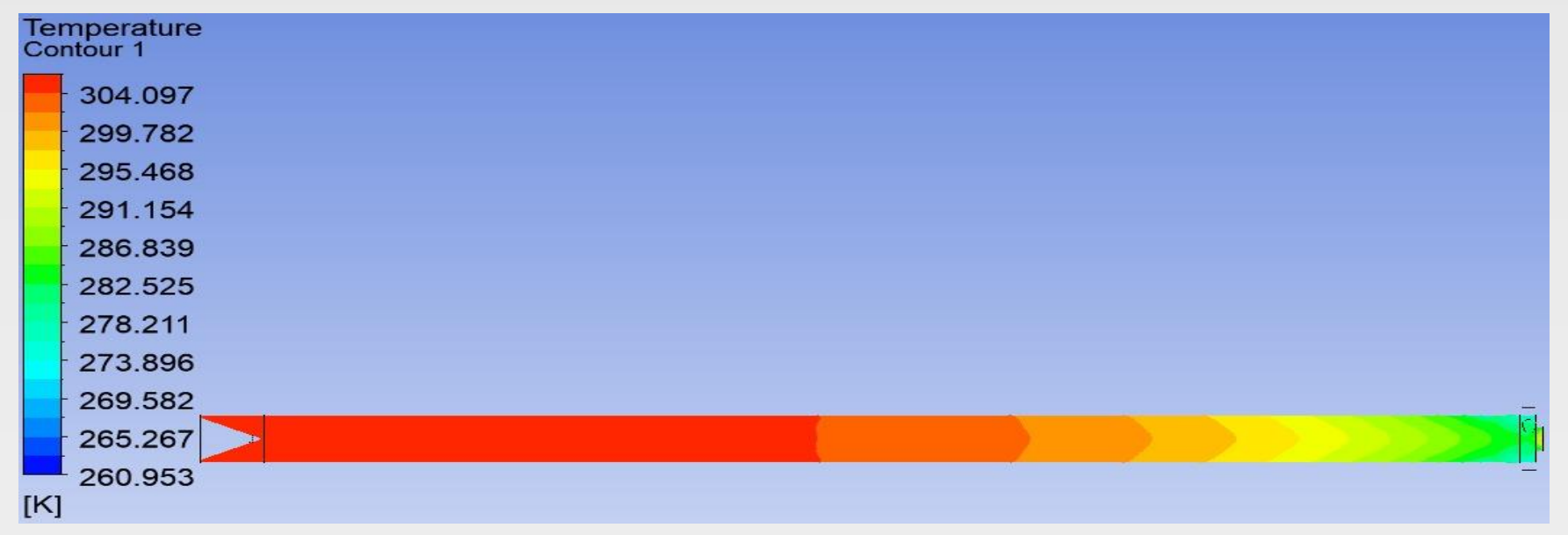


Inlet mesh

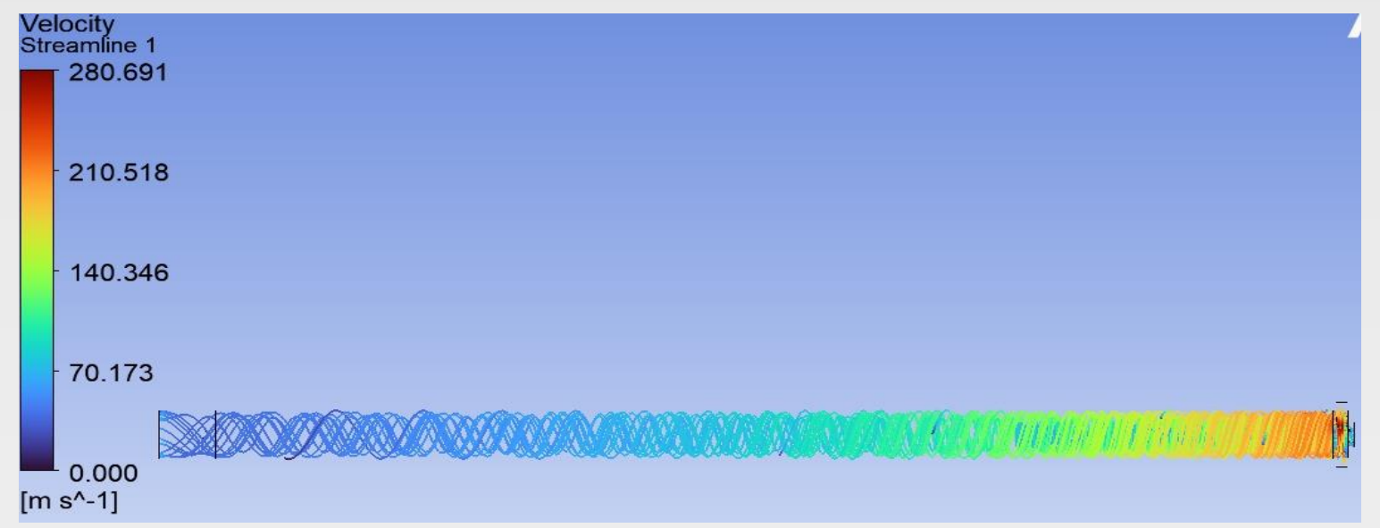
❖ Boundary Condition

Boundary Type	Boundary Conditions	Value
Inlet	Pressure inlet	2×10^5 Pa (Total)
Cold exit	Pressure outlet	101325 Pa (Static)
Hot exit	Pressure outlet	1.19×10^5 Pa (Static)
Walls	Adiabatic with no-slip	Heat flux = 0 $W\ m^{-2}$

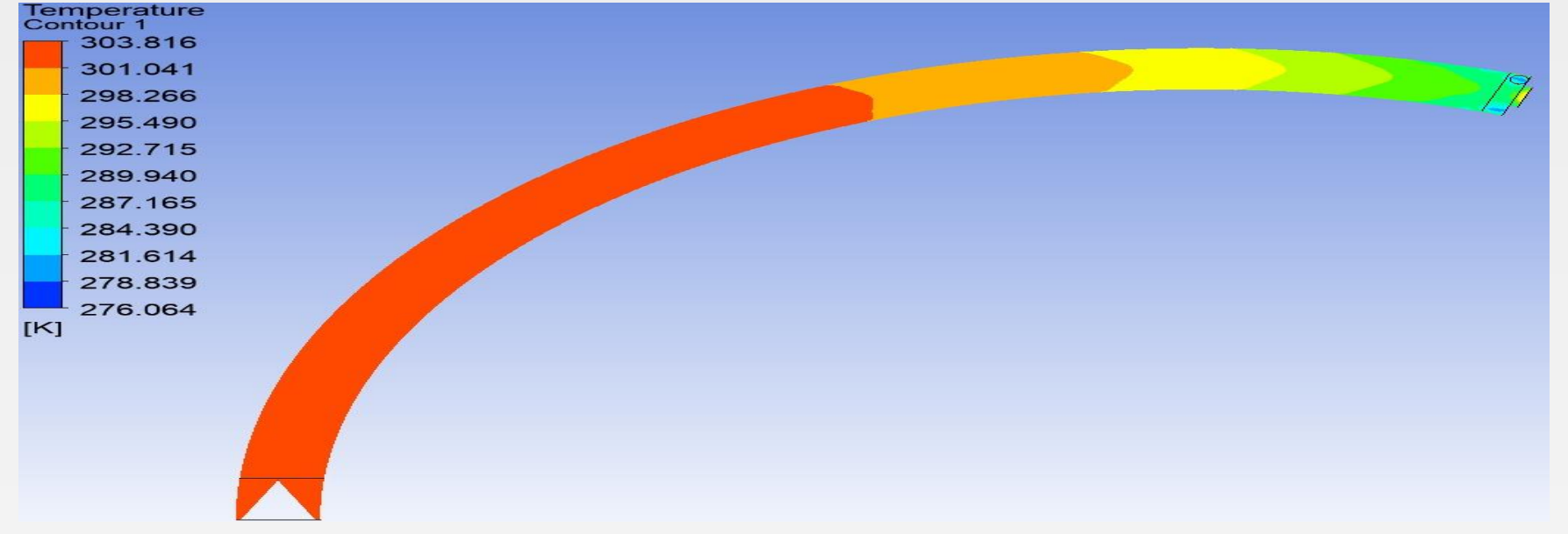
FINITE ELEMENT SIMULATIONS



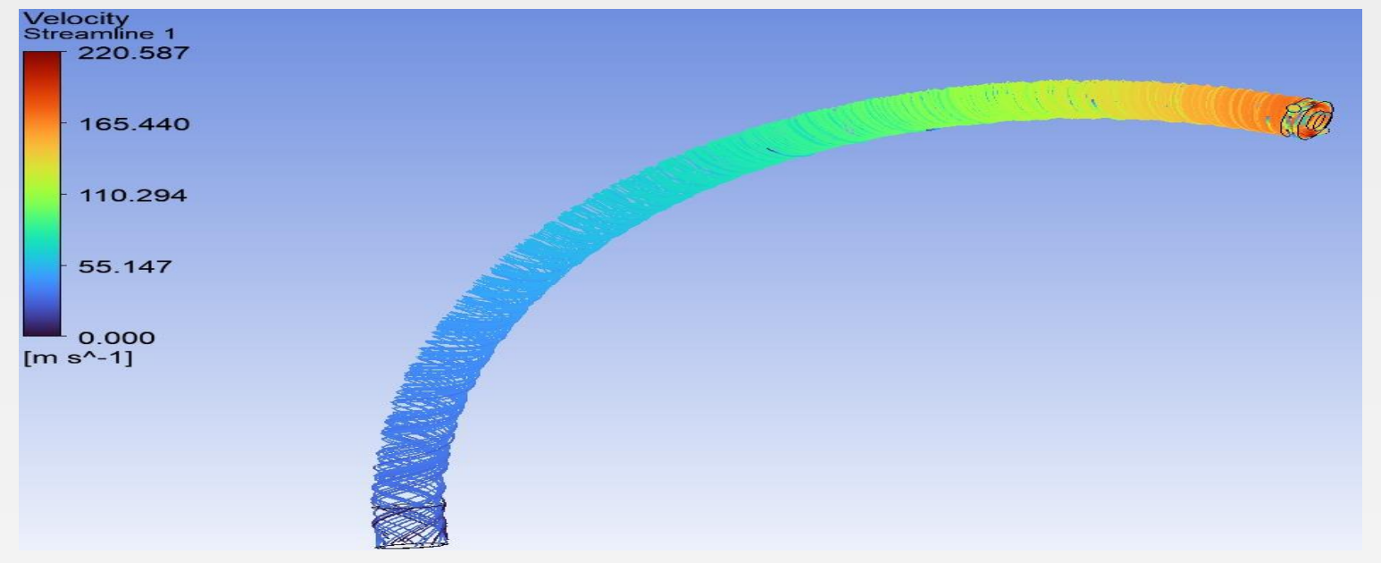
Temperature contour for SVT



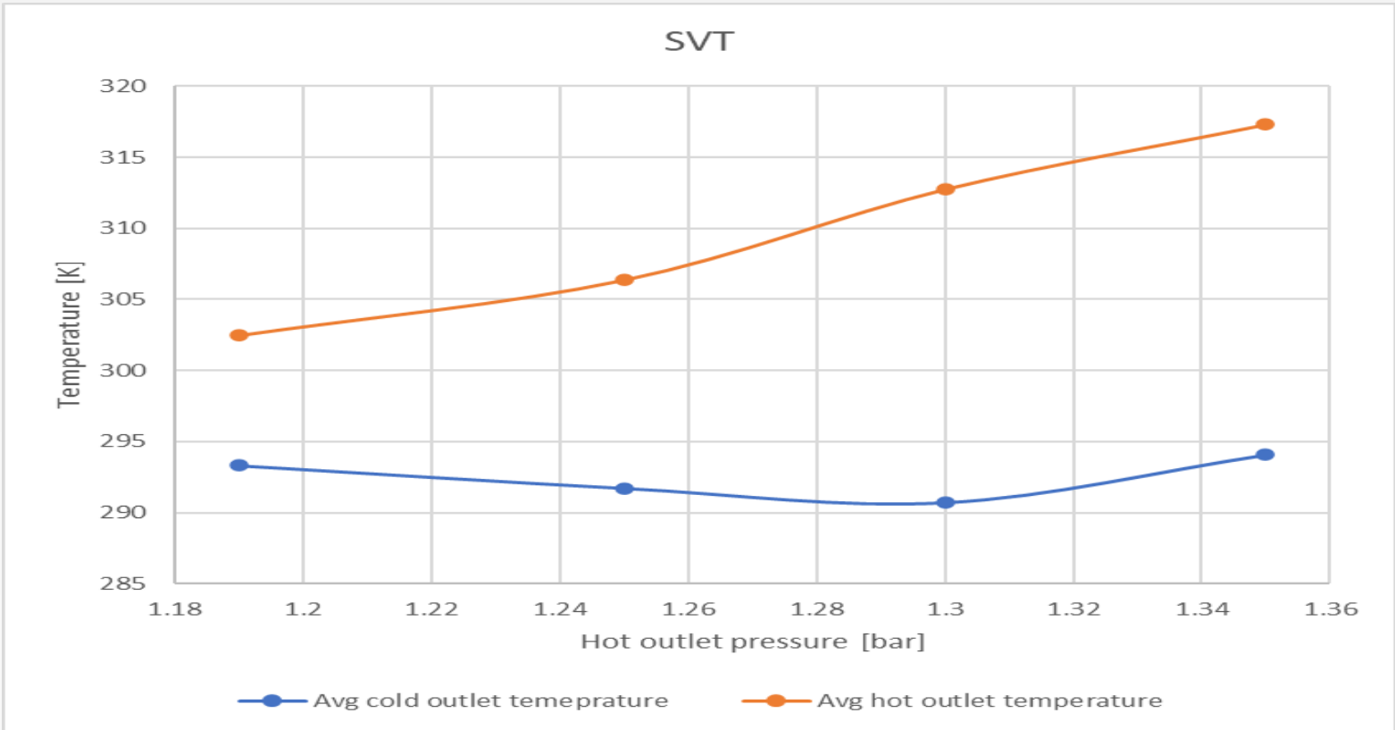
velocity streamline for SVT



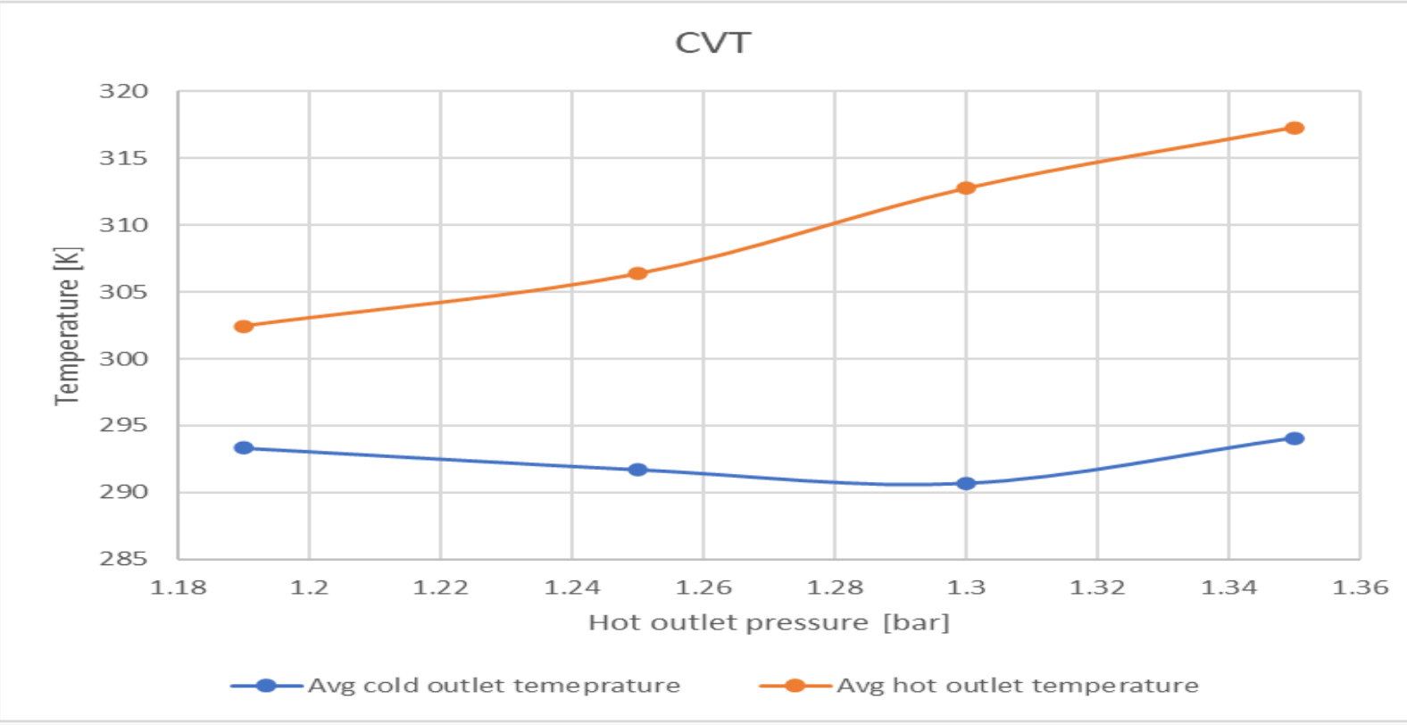
Temperature contour for CVT



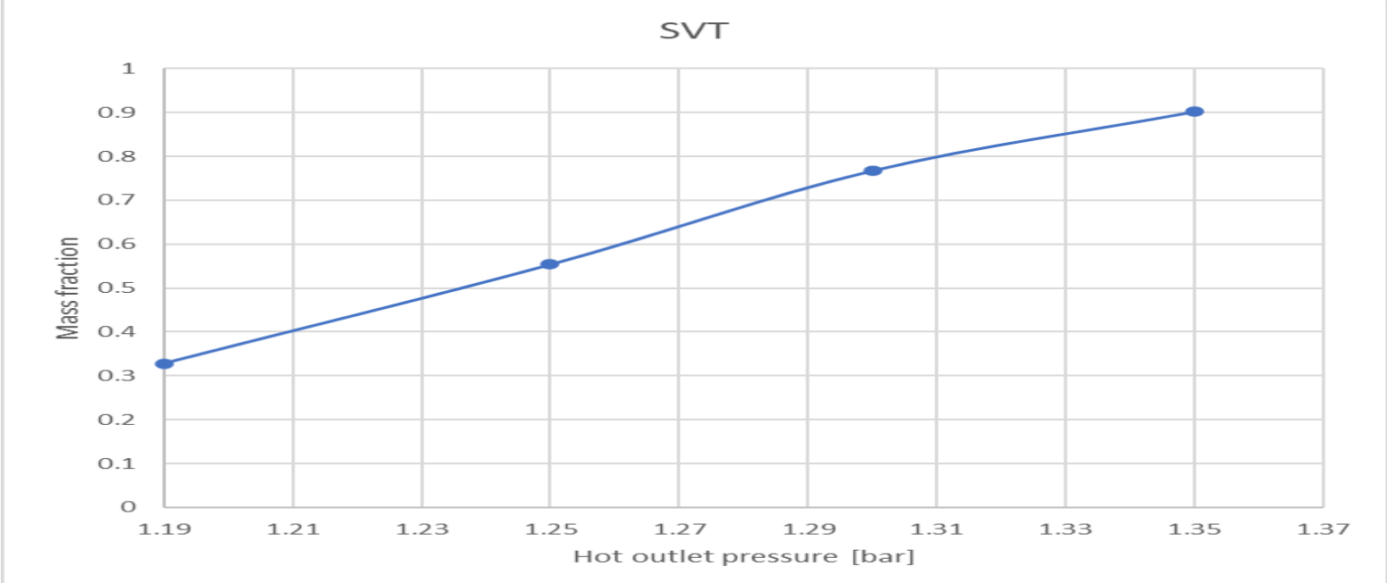
velocity streamline for CVT



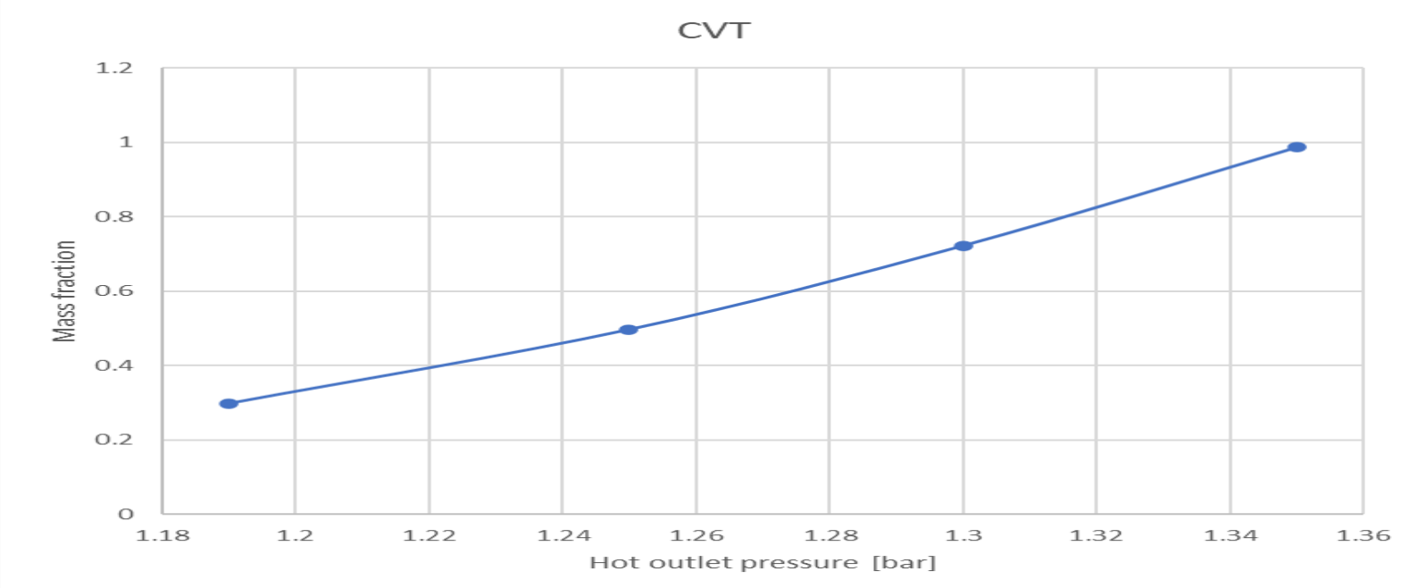
The change of temperatures at both hot outlet and cold outlet for the SVT due to the change in the hot outlet pressure



The change of temperatures at both hot outlet and cold outlet for the CVT due to the change in the hot outlet pressure



The effect of the hot outlet pressure changes on the mass fraction of the cold stream in SVT



The effect of the hot outlet pressure changes on the mass fraction of the cold stream in CVT

CONCLUSIONS

- In conclusion, our comprehensive study on vortex tubes encompassed various key aspects.
- Firstly, the validation process undertaken for both straight and curved vortex tubes demonstrated a good agreement with reference work.
- Secondly, a mesh independence test was conducted to determine the mesh size that give high accuracy with computational efficiency.
- Finally, the thorough examination of the effect of hot outlet pressure change on both SVT and CVT revealed compelling results. Specifically, an increase in hot outlet pressure, induced a noticeable increase in the hot outlet temperature.

REFERENCES

- [1] S. Y. Khan, U. Allauddin, S. M. F. Hasani, R. Khan, and M. Arsalan, “A CFD analysis on the effect of tube curvature, hot flow control valve profile, and inlet swirl on the thermal performance of curved vortex tubes,”
- [2] C. D. Fulton, “Ranque’s tube,” J. Am. Soc. Refrig. Eng., vol. 58, pp. 473–479, 1950.