

REFRIGERATION USING VORTEX TUBE

P. K. V. S. Subramanyeswararao

Assistant Professor, Department of Mechanical Engineering, Raghu Engineering College,
Dakamarri, Bheemili-Mandal, Visakhapatnam-District, Andhra Pradesh., India.,
E mail- pkvssr@gmail.com

Abstract: Vortex tube is a simple energy separating device which is compact and simple to produce and to operate. When compressed air flows tangentially into the vortex chamber through inlet nozzles it splits into hot and cold air streams. There are three regions in the tube. In region one fluid enters, second region is cold and third region is hot. To better understand temperature separation a 3D model of vortex tube was designed using Ansys Workbench. The turbulent flow is used for the study. CFD analysis of vortex tube is done to understand the importance of different inlet shapes. FLUENT is used to carry out the CFD analysis.

Key Words: CFD Simulation, Refrigeration, Temperature Separation,, Vortex tube.

1. INTRODUCTION:

When pressurized air is admitted into the vortex tube in tangential direction, it attains high swirl velocity. This motion sets up a vortex type of flow inside the tube. Due to this, one of the streams exiting the tube is at lesser temperature than the other stream. A Vortex tube consists of the following parts:

- Nozzles for inlet
- vortex chamber
- cold-end orifice
- valve for controlling the flow
- hot end orifice

The vortex tube principle can be used in many applications.

Vortex tube

- Requires little maintenance
- No moving parts.
- Having low thermal efficiency.

In this study numerical method is employed to know the factors effecting vortex tube performance.

Due to reversed flow heat transfer takes place from inner core to outer core

The cold flow exits through the outlet orifice.

The hot flow exits through the annular space between the tube wall and the conical valve at the hot outlet

Hilsch[1] found that the heat transfer due to shear from inner to outer fluid layers is the cause of temperature separation.

Ranque[2] found that the work transfer caused by expansion and compression effects is the main cause for separation of temperature

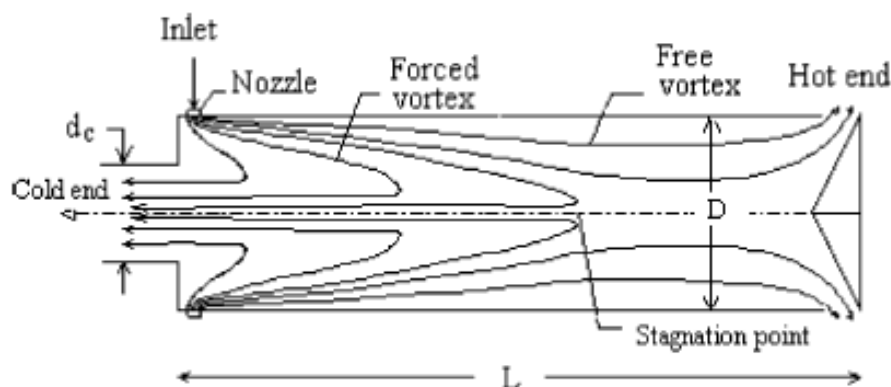


Figure 1. Flow pattern in vortex tube

Ahlborn and Groves proposed that the embedded secondary circulation is the phenomena that cause the energy separation. They measured the axial and azimuthal velocity components to reach this conclusion. Ranque gj” method and apparatus for obtaining from a fluid under pressure two outputs of fluid at different temperatures”(1934)

1.1. Need of the study:

Vortex tube requires less maintenance for getting refrigeration and no moving parts, So it is essential to know the temperature variation due to vortex creation and the dependence of temperature on geometrical parameters

1.2. Scope of study:

- CFD analysis is carried out by varying the geometrical parameters

1.3. Objectives of the study:

- To Study the effects of variation of inlet geometry on the temperature variation

2. CFD METHODOLOGY:

Complete CFD Analysis:

The complete CFD analysis procedure can be divided into the following six stages:

1. First stage: The data required for the simulation are geometry details, fluid properties, flow specifications, initial and boundary conditions
2. Geometry creation: The geometry is created using ANSYS Workbench.
3. Mesh generation: In this stage the flow domain is divided into fine mesh. Variables are accurately calculated on a fine mesh. A fine mesh requires more computational power and time..
4. Specification of Flow Parameters
 - fluid properties,
 - flow models
 - initial flow conditions
 - boundary conditions
5. Calculation of the numerical solution: Iterative calculations is done to arrive at a solution to the numerical equations representing the flow
6. Analysis of Results: The user analyzes the results for checking the results are satisfactory.

3. COMPUTATIONAL MODEL FOR VORTEX TUBE:

1. Problem Description:

Studying the variation of the temperature in the model by varying the geometric parameters

Assumptions:

- Fluid flow is in single phase.
- Inside surface is assumed to be smooth.

2. Method of Solution:

FLUENT software solves governing equations.

SIMPLE algorithm is used.

CFD analysis is carried out in Fluent

3. Steps in problem solving

- Creation of geometry model
- 2. Meshing
- 3. Starting of solver
- Checking the grid.
- Selection of solver
- Flow--turbulent
- Specification of material properties
- Specification of boundary conditions
- Specification of solution control parameter
- Flow initialization
- Calculation of solution.
- Examination of results

(A). *Geometry*: Geometry is created in ANSYS workbench.

Various parts:

Part number	Part of the model
1	INLET—Compressed fluid
2	OUTLET-Cold fluid
3	OUTLET-Hot fluid
4	Fluid domain

Table-1 Various parts

Models created:

- Model –I Inlet circular cross section
- Model –II Inlet rectangular cross section
- Model –III Inlet trapezoidal cross section
- Number of nozzles=6

Parameter	Circular Cross section	Rectangle Crosssection	Trapezoidal Crosssection
Diameter of vortex tube (D)	12 mm	12 mm	12 mm
Inlet Nozzle	Diameter=6mm Depth=7mm	Length=3mm Width=1mm Depth=7mm	Top length=2mm Bottom length =1mm Width=3mm Depth=7mm
Diameter of cold end (d)	Diameter =6mm	Diameter =6mm	Diameter =6mm
Length of vortex tube (L)	106 mm	106 mm	106 mm
Valve Semi- cone angle (α)	30°	30°	30°

Table-2 Geometrical parameters

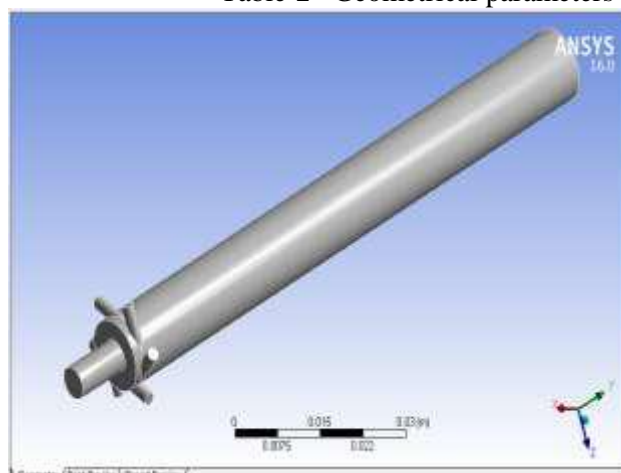


Figure-2 Geometry of Inlet circular cross section

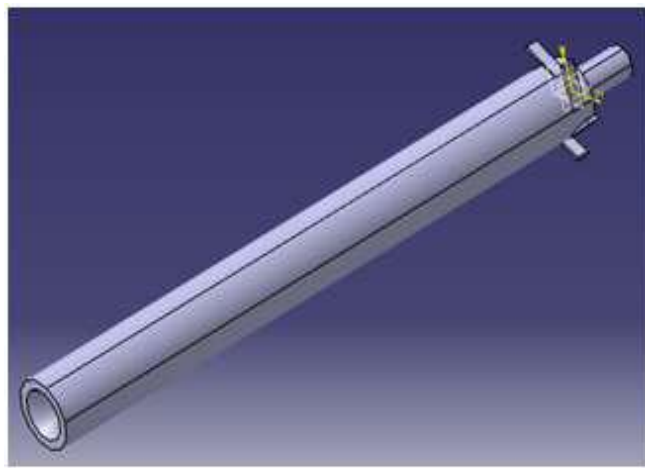
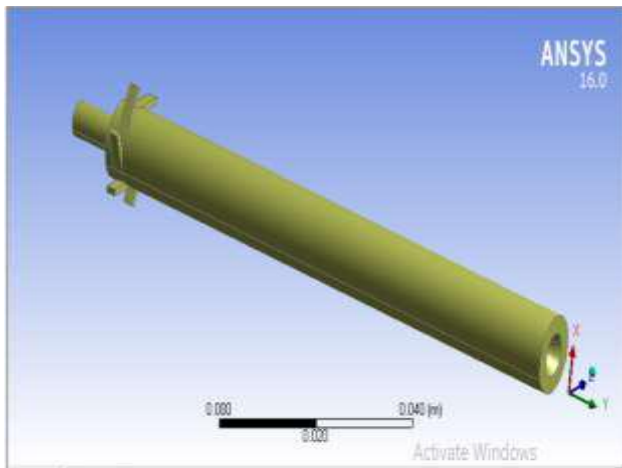


Figure 3 Geometry of Inlet Rectangular cross section (B) Meshing:

Figure 4 Geometry Inlet trapezoidal cross section

Meshing was done in Ansys Workbench Meshing software. The inlet, cold outlet, hot outlet vertices were marked as named sections.

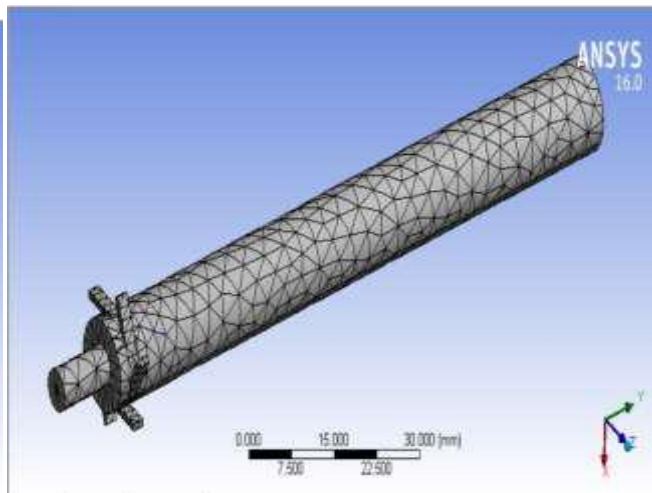
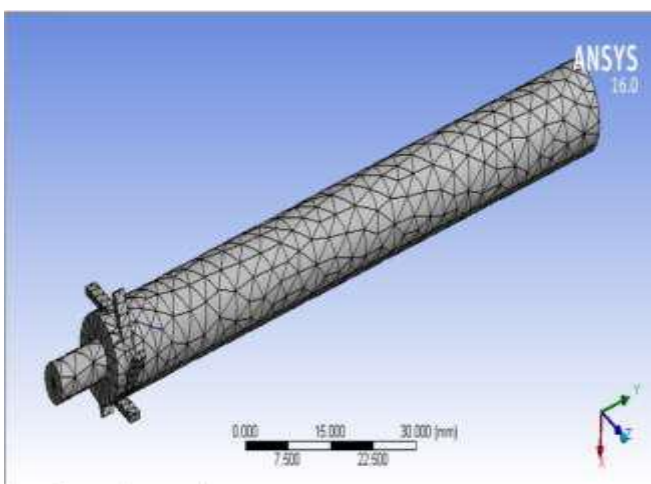


Figure 5 Meshing of Inlet circular cross section

Figure-6 Meshing of Inlet Rectangular cross section

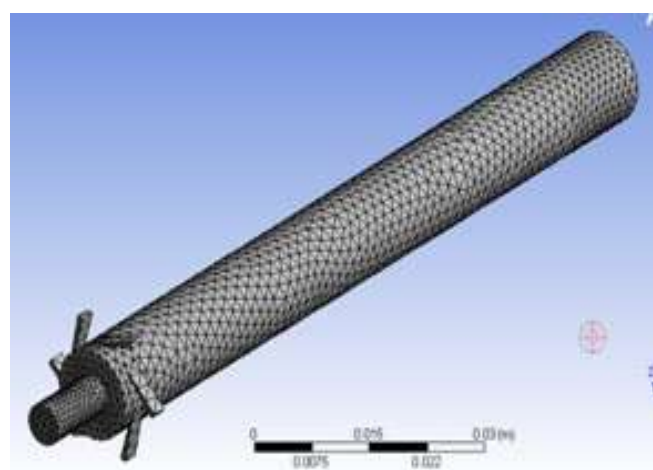


Figure-7 Meshing of Inlet Trapezoidal cross section

Ansys Fluent16
 Launching ANSYSFluent.
 Dimension - 3D, Double Precision,
 Processing - Serial type
 OK.

(C). Setup for solution

Checking the mesh

1. General:

For simulating this geometry, real life assumptions were made.

The general setup is as under:

- Solver type was selected as to be Pressure Based
- 3D space model was selected
- Time was selected to be steady
- Velocity formulation was considered absolute

2 Solution Models:

Energy - ON

Viscous model --k-ε model (2 equations).

3 Materials:

MATERIAL PROPERTIES:

AIR	
Density	1.225 Kg/m ³
Specific heat	1006.43 j/kg-k
Thermal conductivity	0.0242 W/mK.
Viscosity	1.7894 kg/m-s
Reference temperature	298.15k

Table-3 Material Properties

4. Cell Zone Conditions:

FLUID-AIR

5. Boundary Conditions:

Inlet:

Velocity-Inlet=2m/s

Total temperature= 290

Total Pressure=0

Turbulent intensity=5%

Turbulent viscosity ratio=10

Cold Outlet:

Pressure Outlet, Gauge Pressure=0

Hot Outlet:

Pressure Outlet, Gauge Pressure=0

Turbulent intensity=5%, Turbulent viscosity ratio=10.

No slip boundary condition at wall surface.

The wall --assumed to be stationary.

6. Solution:

i. Solution Methods:

- Simple Scheme
- Least Square Gradient
- Standard- Pressure
- Second Order for Upwind Momentum
- Second Order Upwind for Turbulent Kinetic Energy
- Second Order Upwind for Turbulent Dissipation Rate

ii. Solution Control:

Under relaxation factors

- Turbulence Kinetic energy= 0.8
- Turbulence Dissipation Rate= 0.8
- Turbulence Dissipation Rate= 0.8

iii. Monitors:

Convergence Criteria – Residuals - 0.001

iv. Reference Values

- v. Solution Initialization
Initialization of Solution
- vi. Solution Control and Initialization:
the solution initialization method -Hybrid Initialization
- vii. Measure of Convergence –good convergence to get accurate result.
- viii. Run Calculation
The number of iteration is set to 500
- ix. The solution is calculated
- x. Results- obtaining various contours, vectors and plots

4. RESULTS AND DISCUSSION:

Convergence of Simulation-

The convergence is required to get the parameters

The higher pressure at the wall is due to radial pressure gradient.

The hot flow at higher temperature comes out from the warm outlet.

The cold flow at lower temperature comes out from the cold outlet .

Trapezoidal model gives the lowest cold outlet temperature.

It is evident from the temperature contour, cold temperature can be obtained by varying geometric parameters.

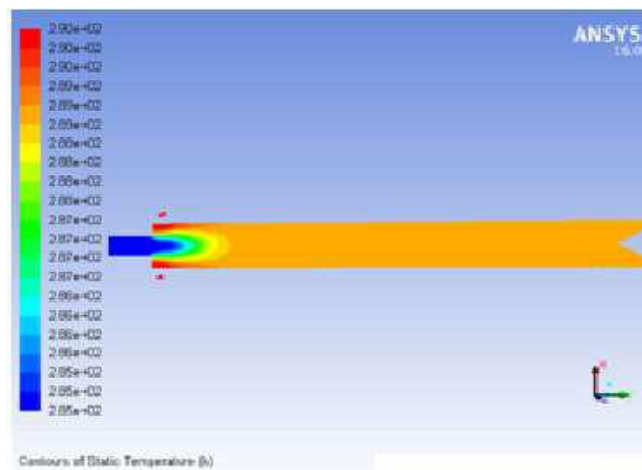


Figure 8 Distribution of Total temperature in the vortex tube for Circular cross-section

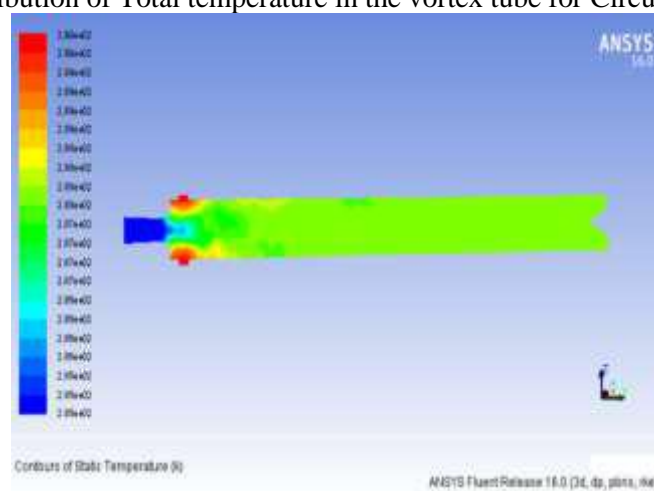


Figure 9. Distribution of Total temperature in the vortex tube for Rectangular cross-section

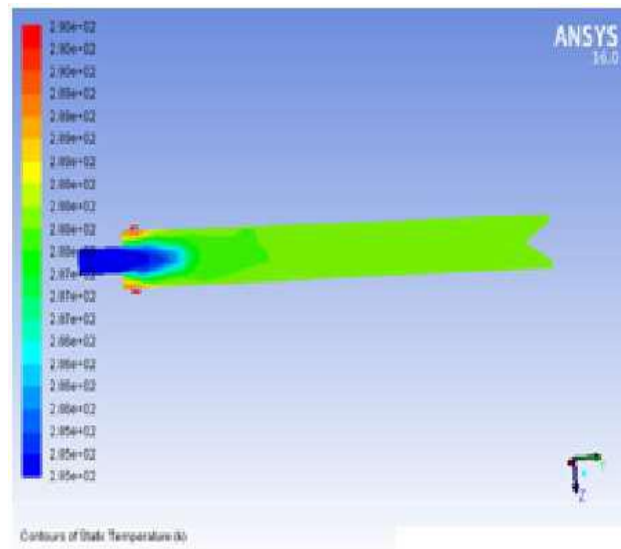


Figure 10. Distribution of Total temperature in the vortex tube for trapezoidal cross-section
CONTOURS OF VELOCITY MAGNITUDE:

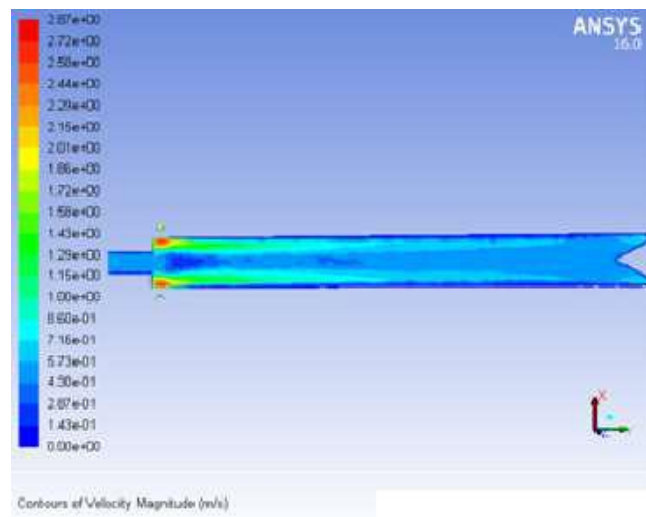


Figure11 Velocity Magnitude Contours for circular cross section

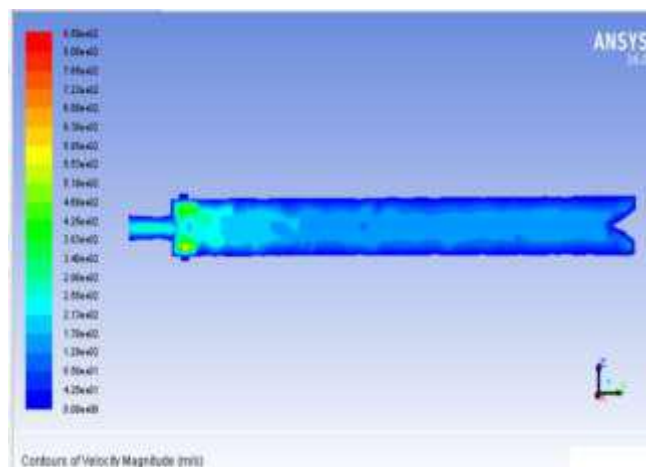


Figure 12. Velocity Magnitude Contours for Rectangular cross section

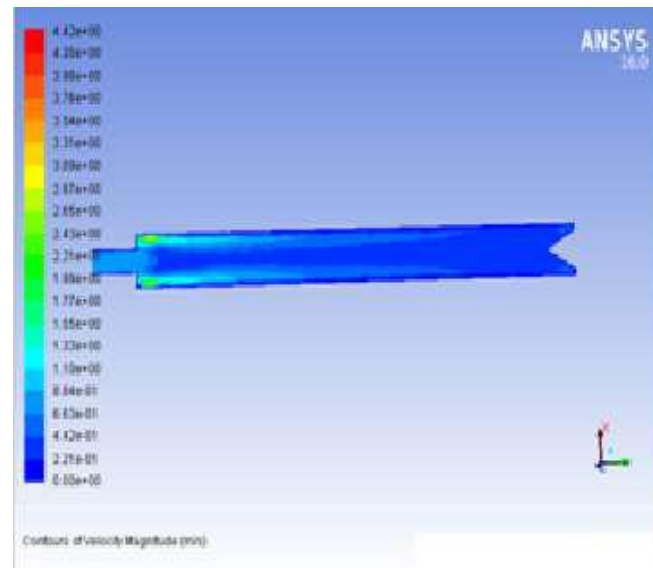


Figure 13. Velocity Magnitude Contours for trapezoidal cross section

CONTOURS OF STATIC PRESSURE

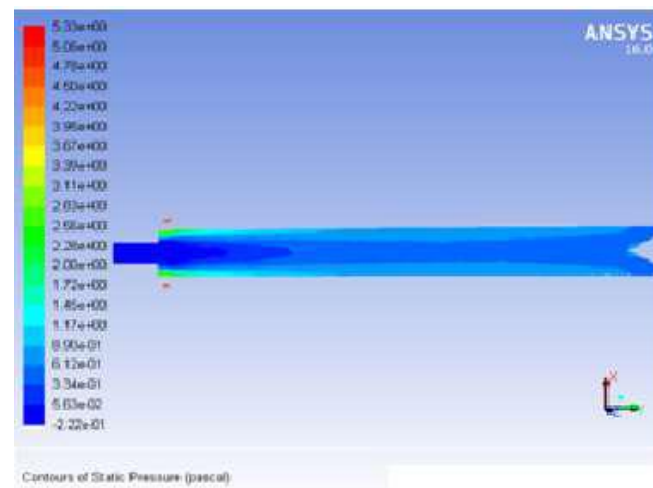


Figure 14. Static pressure Contours for circular cross section

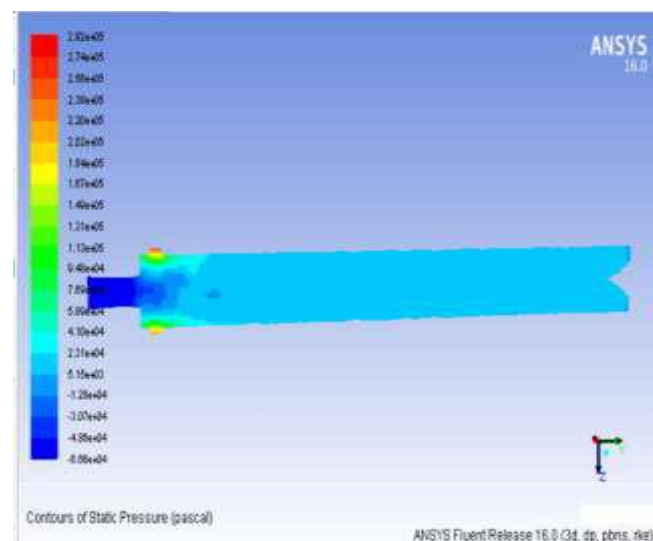


Figure 15. Static pressure Contours for rectangular cross section

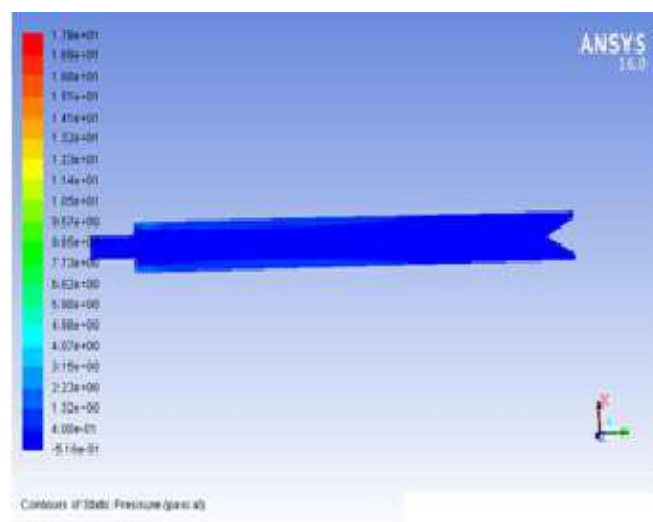


Figure 16 Static pressure Contours for trapezoidal cross section

RESULTS AND COMPARISON

MODEL	FLUID	INLET TEMPERATURE K	COLD OUTLET TEMPERATURE K	HOT OUTLET TEMPERATURE K
CIRCULAR	AIR	290	287	288.92047
RECTANGULAR	AIR	290	286	288.14719
TRAPEZOIDAL	AIR	290	285	287.75549

Table 4 Results and comparison

5. CONCLUSION AND FUTURE WORK:

Conclusion:

3-Dimensional steady state analysis have been conducted.

Temperature separation is possible when compressed fluid enters the tubes tangentially.

Trapezoidal model gives the lowest cold outlet temperature. when a fluid is tangentially introduced into the vortex tube, it separates in to two streams cold and hot As the fluid moves axially the velocity decreases and reaches a stagnation point The vortex tube performance is dependent on compressed air properties

Scope for future work:

Research in Vortex Tube Refrigeration by simulating flow of various refrigerants to get lower temperature.

Pressurized gases exiting as waste can be used in Vortex tube for recovering the pressure energy.

Material of the vortex tube--the non metallic materials like homogeneous wood, Acrylic, Nylon, PVC, CPCV etc, may be varied to get lower temperature

REFERENCES:

1. R. Hilsch, "The use of the expansion of gases in a centrifugal field as cooling process," Review of Scientific Instruments, vol. 18, no. 2, pp. 108–113, 1947.
2. Ranque, "Experiments on expansion in vortex with simultaneous exhaust of hot air and cold air". Le journal de Physique et le Raiuum(Paris)pp 112-114(1965).