

3D Simulation of Flow Inside a Counter Flow Vortex Tube

Prajit Dhakal

Undergraduate Student, Aerospace Engineering,
Pulchowk Campus, IOE, Tribhuvan University

Abstract

The objective of this study was to develop and computational environment inside *OpenFOAM* for simulating flow inside the RHVT and perform different analysis. The project was setup from the existing tutorial cases inside *OpenFOAM* repository and needed steps for its setup are explained in detail. The geometry for the problem is given below in figure [1] which was generated using *FreeCAD* with some modification and mesh generation was done using *snappyHexMesh* utility. Pressure based solver for compressible and turbulent flow were used i.e. *rhoSimpleFoam* which is steady state solver and *sonicFoam* which is transient solver. *rhoSimpleFoam* was discarded later and only *sonicFoam* was used due to reverse flow in steady state solver. The obtained results were compared and validated to the available reference studies. Details regarding geometry and flow has been listed in Table[1].

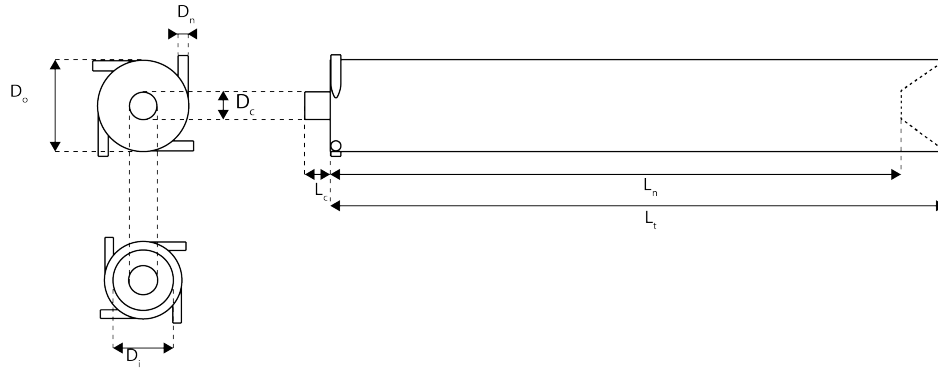


Figure 1: Geometry Details of Computational Domain

Parameter	Value
Inlet Nozzle Diameter (D_n)	1.6 mm
Cold Outlet Diameter (D_c)	5.43 mm
Tube Outer Diameter (D_o)	14.48 mm
Hot Outlet Internal Diameter (D_i)	11.28 mm
Cold Outlet Length (L_c)	4 mm
Tube Length (L_t)	97 mm
Hot Outlet Nozzle Length (L_n)	90 mm
Inlet Velocity (U_{in})	200 m/s
Inlet Temperature (T_{in})	300 K
Hot Outlet Pressure (P_{hot})	1.15×10^5 Pa
Cold Outlet Pressure (P_{cold})	1.00×10^5 Pa

Table 1: Geometric and Flow Parameters of Vortex Tube