CFD study of deLaval nozzle with Fanno and Rayleigh condition

Abstract-Computational fluid dynamics (**CFD**) is a branch of fluid mechanics that uses numerical analysis and algorithms to solve fluid flow problems. The objective of the current study is to study the variation of flow parameters like pressure, temperature and Mach number by adding friction & heat transfer to a converging-diverging nozzle with the help of CFD. At first, a comparative study is done with an already published paper (the only trend of the graph is matched in the subsonic regime) and then a new contribution has been added. The mesh is done using the *blockMesh* utility of OpenFoam. Also, this paper describes the implementation of heat-flux and friction in OpenFoam. Widely used k-epsilon turbulence model with wall function treatment at wall boundary used. Discretized conservation equations like continuity, momentum, energy and turbulence are solved simultaneously using CFD Open Source package **OpenFOAM® V-7.0** with *rhoCentralFoam* solver to simulate the flow Physics.

Problem statement

In this case, to test the validity of our solver, we first validated against [1] the constant pipe case. Then we performed case studies on our own geometry(Fig:1) and also compared the trends of the graph with the nozzle flow of paper for the subsonic condition. After a reasonable agreement between the two, we went forward and extended our case to supersonic flow and supersonic flow with shock.

A convergent-divergent nozzle of geometry as shown in fig. 1 is considered For this case The nozzle cross-section varies as -

$$A(x) = \begin{cases} 1.75 - 0.75\cos(0.2x - 1)\pi, & 0 < x \leq 5 \\ 1.25 - 0.25\cos(0.2x - 1)\pi, & 5 < x \leq 10 \end{cases}$$

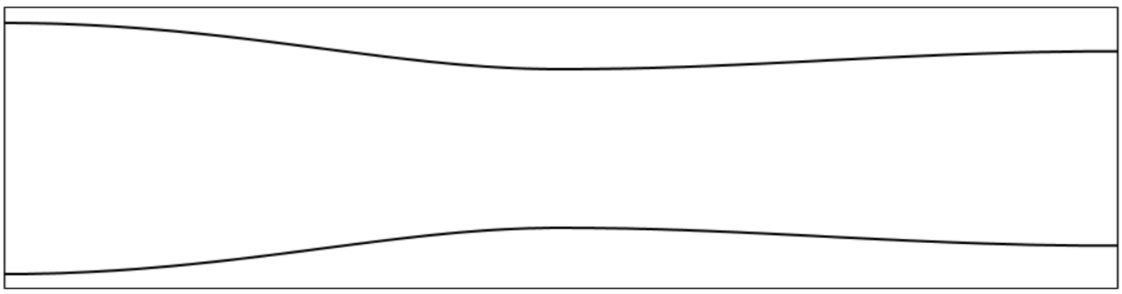

Fig: 1 The configuration of flow through a convergent-divergent nozzle

- ❖ For this case, the value of the **darcy friction factor(f)** is **0.05**.
- ❖ And, the value of the **heat flux** is **5000 W/** m^2 .

Reference :

[1] Bandyopadhyay, Alak, and Alok Majumdar. "Modeling of Compressible Flow with Friction and Heat Transfer using the Generalized Fluid System Simulation Program (GFSSP)." Thermal Fluid Analysis Workshop. Vol. 10 (2007): Web. 20 January 2017.