FACULTY OF MECHANICAL ENGINEERING

BMM4873 COMPUTATIONAL FLUID DYNAMICS

ANALYSIS OF COMPRESSIBLE FLOW THRUGH A CONVERGING-DIVERGING NOZZLE USING CFD

1. Objective

It is known that air flow through a converging-diverging nozzle is one of the most common problems used for modeling compressible flow using computational fluid dynamics.

The purpose of this project is to model and analyze a two dimensional inviscid compressible flow of air through a converging-diverging nozzle by using Computational Fluid Mechanics (CFD) technique and to compare the results with 1D isentropic flow analytical results.

2. Problem Description

For the analysis, consider air flowing at high speed through a convergent-divergent nozzle as shown in **Figure 1**. The stagnation pressure p_o at the inlet is atmospheric pressure (101.325 kPa) and the stagnation temperature T_o at the inlet is 23 ^oC. The static pressure *p* at the exit is 3.7 kPa. The flow is modeled as inviscid flow.

The theory for compressible flow can be obtained from the lecture notes I uploaded in Kalam or from any fluid mechanics book (such as, Fluid Mechanics Fundamentals and Applications 2nd edition by Yunus A. Cengel and John M. Cimbala).

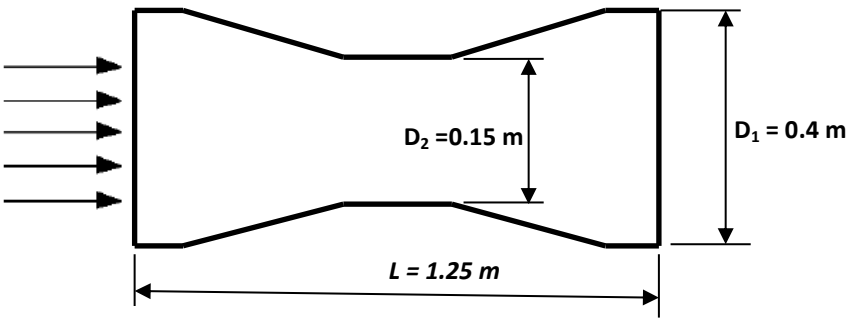

Figure 1. Schematic of flow through a converging-diverging nozzle

3. Expected Results

Your analysis should include the following:

- Velocity, Mack number, pressure and temperature distribution along the nozzle length
- Comparison of Mach number, pressure and temperature between simulation (at the axis) and 1D nozzle flow results.
- Contours of Mach number, pressure and temperature.

4. Report

The report should concentrate **on compressible flow through converging-diverging nozzle, and it should be not more than 10 pages.**

In your report, please include abstract, objective, theory, results and discussion, conclusion, and references (if any).