ME 639 **FLOW SIMULATION, PROJECT-2** **Spring 2024**

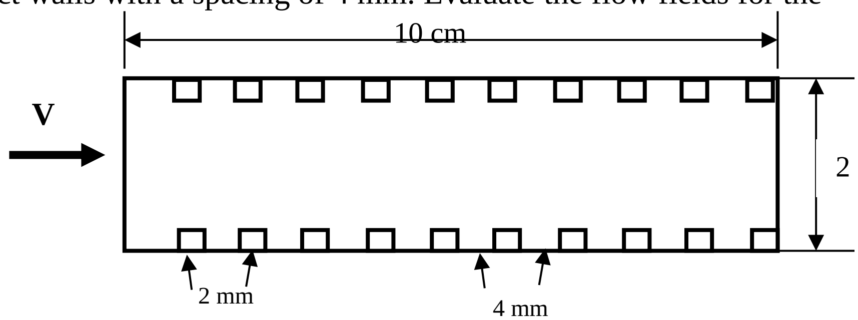
Part 1: Duct Flow

Develop a grid and analyze the developing airflow in a 2 cm wide and 10 cm long channel. a) For an air velocity of 0.1 m/s, evaluate the velocity profile and wall shear stress. (Evaluate the Reynolds number and verify that the flow is laminar). Determine the sensitivity of the solution to the grid size and compare the numerical velocity profile with the exact laminar flow solution for a fully developed steady parallel flow profile. Also, discuss the nature of the developing flow.

b) For an air velocity of 5 m/s, evaluate the velocity profile using the $k-\varepsilon$, and the stress transport models. Discuss the difference between laminar and turbulent flows. Find the friction coefficient for the air velocity of 0.1, 0.5, 5, and 10 m/s and compare the results with the moody diagram or empirical equations.

a) For smooth sidewalls, evaluate the velocity profile and turbulence intensity using the $k-\varepsilon$, and the stress transport model. Discuss the difference between the various turbulence models.

b) Modify the geometry of the duct and assume that there are rectangular roughness elements 2 mm wide and mounted on the duct walls with a spacing of 4 mm. Evaluate the flow fields for the inlet velocities of 0.5 and 5 m/s. Plot the velocity vector field and the velocity magnitude, pressure, and turbulence intensity contours. Also, plot the streamlines. Find the pressure drop for a range of velocities between 1 to 10 m/s and compare the friction coefficient with the empirical equations.



c) Repeat part (b) when only the upper wall is rough. Again, compute the pressure drop and friction factor and compare the results with the empirical equations. Also, compare the flow profiles and turbulence intensity with the experimental data of Launder et al. (1975).

Part 2: Comparison with the Experimental Data

For one of the cases given in the notes on turbulence modeling or a paper in the literature for which experimental data are available, evaluate the flow field using the $k-\varepsilon$, $k-\omega$, and the stress transport turbulence models and compare the result with the experimental data. Discuss the accuracy of different turbulence models.

Part 3: Thesis

If you are already familiar with Ansys-Fluent, instead of Parts 1 and 2, you can simulate a flow field related to your thesis with a comparable effort. Specific projects should be discussed with the course instructor.

Reports and Due Dates: The project reports should include an abstract, introduction, results, figures with captions, a discussion of the results, conclusions, and references. The PDF copy of the report and copies of the case and data files should be uploaded to Moodle by **April 23, 2024.**