ME3700 – Fall 2003
Fluent Project – Laminar Boundary Layer Computation
Due December 4, 2003

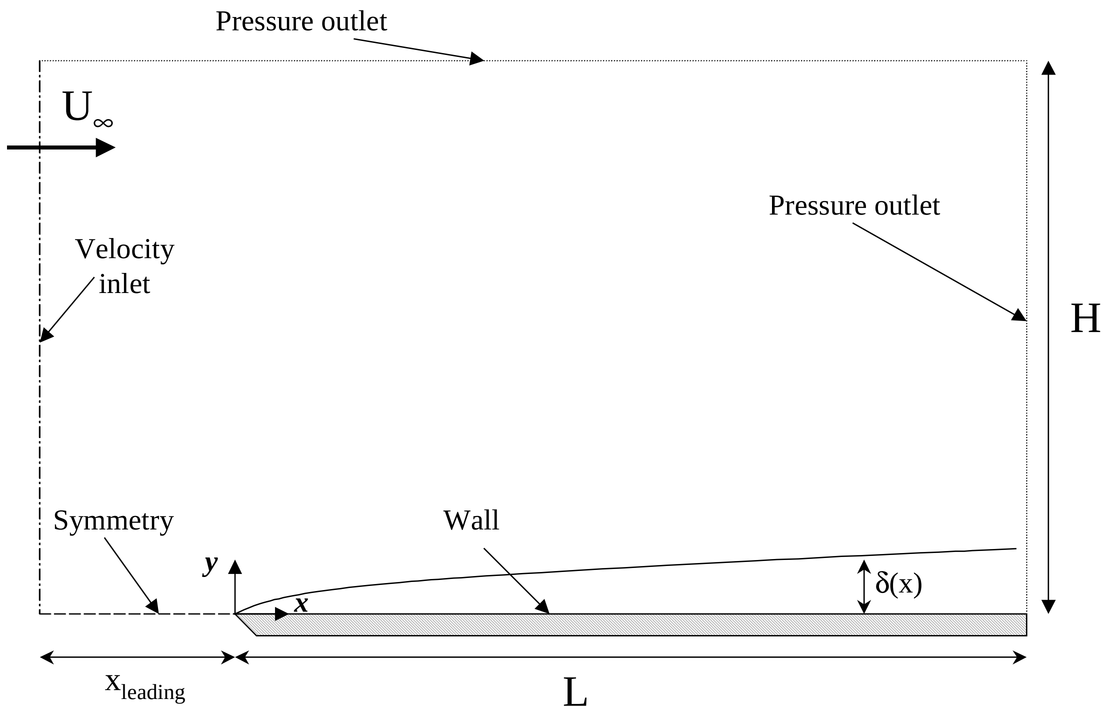

Figure 1 – Schematic of the computational domain and boundary conditions for laminar flow over a flat plate.

The purpose of this Fluent project is to introduce you to “canned” Computation Fluid Dynamics (CFD) codes within the framework of boundary layer flows. Fluent is widely used throughout industry and is a powerful tool, however as you will find as you proceed with this project, you will see that running a Navier-Stokes solver in complex geometries (and not so complex geometries) can be difficult. You will find that your fundamental understanding of boundary layer flows is necessary to interpret and understand the results. Treating a tool like Fluent as a “black box” can be very dangerous, and one should never forget that the assumptions and simplifications along with other issues such as “grid resolution” and numerical approximations that could lead to incorrect solutions.

In the accompanying figure, *x* is the freestream flow direction, parallel to the flat plate. Coordinate *y* lies normal to the plate. The *z*-direction is out of the page. The problem is 2D so the plate extends to infinity in the *z* direction. However, for purposes of calculating drag coefficients, etc., define *b* as the width (or depth) of the plate in the *z* direction. *b* is generally taken to be one unit length (1 m) for convenience, and drag is reported as drag per unit span (or drag per unit depth) in units of N/m (e.g., F_D/b).

The length of the plate is L=0.5m. One of the first issues to deal with is the domain height, H. You must pick H, such that it is "far enough away" from the boundary so as not to impact the solution near the wall, but not so far away as to waste time.

Two commercial software codes will be needed for this project. Gambit will be used to generate a "mesh" or grid upon which calculations will be made. It will also be used to prescribe the boundary conditions for the problem. Fluent is the actual fluid flow solver.

Problem:

1) Set up and run the Gambit and Fluent Modules accompanying the assignment. We will choose water as our working fluid and compare the results to our air flow experiments in the laboratory.
2) Include the following Fluent plots: Final Grid used (include a note on what the resolution was and how different resolutions effected the flow). What H did you choose, why? The velocity profiles at locations x=0.1, 0.30 and 0.5. Plot contours of u-velocity and v-velocity. How many iterations did it take to get your solution?
3) Plot the non-dimensional Fluent results against your laboratory experimental data and the Blasius solution and compare them (i.e., y/δ vs u/U_∞).
4) Compare the drag results (F_D/b) with the Blasius solution in the textbook.
5) Calculate $\delta=\delta_{99}$, δ^*/δ and θ/δ (for all 3 locations) and compare with the experimental data and the Basius solution.
6) What equations were solved in running this simulation? What assumptions allowed for this?

In all of the above questions discuss what might be causing discrepancies between the different methods for obtaining the boundary layer data.