Project Part II

MAE 6220 Due: 11/17/17 @5pm by email

In this project, we are investigating laminar flow over a smooth 2-D cylinder. The domain size and other relevant flow properties are given in Figure 1. You are asked to investigate the sensitivity of the solution to the computational grid. This is usually called a grid independence study and involves solving the same problem on a sequence of gradually refined girds. Start with a coarse mesh, then redo the calculation using a finer mesh, etc. Continue to refine the mesh until flow features (e.g. drag coefficient) do not change with respect to the mesh size. Note that the finer and coarsest grids you consider should differ by at least a factor of 4.

In order to calculate drag, you can add a drag monitor in the 'Monitors' section of FLUENT. Prescribe the appropriate reference values, convergence criteria and boundary conditions. Note that the upper and lower horizontal boundaries are slip-walls meaning that shear at the wall is zero and this has to be taken into account when prescribing boundary conditions.

Prepare a report that includes the following:

1- Give an overall description of the flow based on the behavior of contours of velocity (u and v) and the pressure, as well as, flow streamlines.
2- Compare the drag coefficient, $C_D = \frac{D}{\frac{1}{2}\rho U^2 A}$, you computed to reference results (see [1]).
3- Compare the angle at which the flow separates at with reference results in [1].
4- Present mesh statistics and features. Explain why you chose that particular grid and why you think it is suitable for this problem.
5- Increase the incoming velocity in a way that the Reynolds number increases to 100. What are the differences you observe in the flow? Are you results in agreement with the reference results reported in [1]?

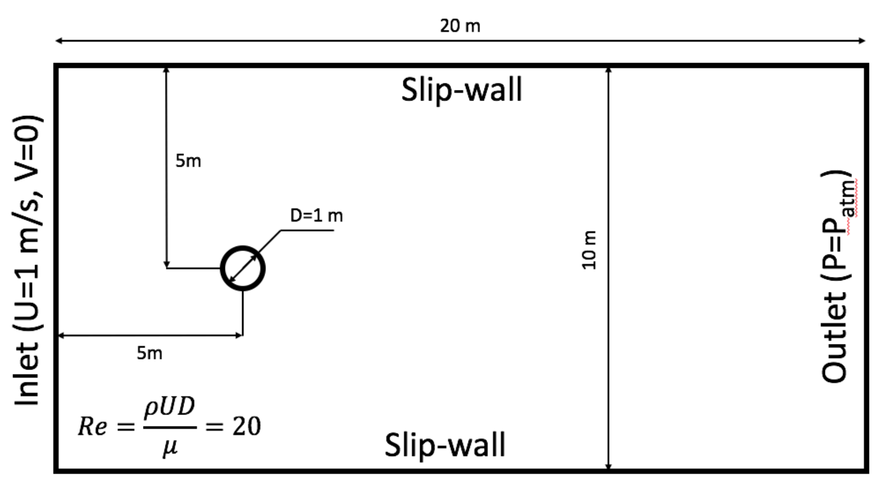

Figure 1: Flow past a circular cylinder, a sketch of the computational domain and flow parameters.

[1] Dennis, S., & Chang, G. (1970). Numerical solutions for steady flow past a circular cylinder at Reynolds numbers up to 100. *Journal of Fluid Mechanics, 42*(3), 471-489. doi:10.1017/S0022112070001428