MAE 423 F19
Heat Transfer

Final Project

Date: Mon, 4 Nov ‘19
Due: Mon, 13 Jan ’20 5pm EST (day before Dean’s Date)

1. The goal of the project is to numerically simulate two-dimensional unsteady heat convection from bodies of arbitrary geometry. This involves the simulation of unsteady incompressible viscous flows with heat transfer at relatively low Reynolds numbers where three-dimensional effects may be negligible. Although this project can readily be accomplished by each student working alone, it is permissible to form a small group (no more than three students) and turn in a single final report, with each student’s work and responsibilities clearly delineated. Of course, it will be expected that the results and output (i.e. project report) will be somewhat proportional to the number of student(s) in the ‘group.’

2. The flows are to be simulated numerically using the “Streamfunction-Vorticity” (ψ-ω) method, which will be presented and discussed in class. The velocity field u,v(x,y) and vorticity field ω(x,y) are immediate outcomes of the simulation, and are used as inputs to the unsteady energy equation, in finite-difference form, to explicitly advect and diffuse the heat.

3. You may use whatever programming language with which you are comfortable. However, MATLAB or C should be seriously considered. MATLAB provides a suite of useful tools, is readily adaptable to array manipulation, and is relatively easy to debug, although the normal uncompiled interpreter generally executes the program rather slowly in comparison to C, which generates efficient code that will that will run faster, but with just a bit more programming effort.

4. For simplicity use, or at least start your code development with, a square mesh, where the grid spacing $\Delta x = \Delta y = h$. Most of the problems may be adequately solved using a 200 x 500 grid-point mesh.

5. Suggested boundary conditions for an external flow are:
    a. Uniform upstream inflow
    b. Constant vorticity and constant velocity outflow downstream
    c. Parallel flow (v = 0) at the top and bottom mesh surfaces

6. Utilize the inviscid solution for the streamfunction ψ as the initial condition for the viscous simulation. The inviscid solution is found from solving $\nabla^2\psi = 0$ iteratively for the interior points in the grid. Once the solution has converged, the streamfunction-vorticity method is then used to determine the velocity components, u and v, in the convective flow, which subsequently advances the vorticity, along with the thermal field.

7. Results should be presented graphically:
    a. Plot velocity and temperature distributions normal to surfaces where appropriate (such as at several locations along the boundary: think of the velocity u(y) and T(y) plots we often sketch in class).

b. Map the streamfunction and temperature to pixels with either assigned grey-levels or color-values to ‘visualize’ the entire domain at various points in time.
c. Create movies which depict temporal flow evolution.

A more detailed discussion of the format of the final report will be presented.

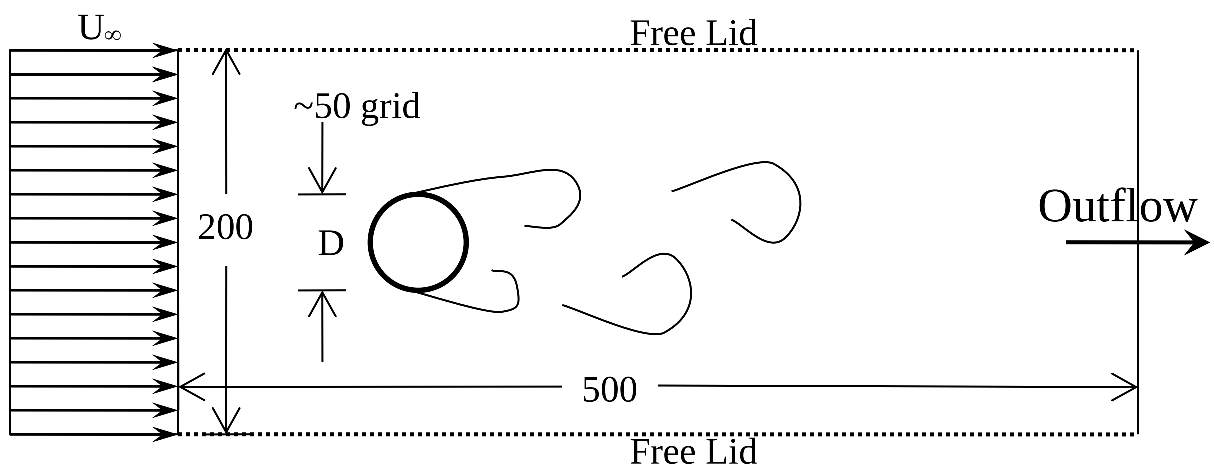
8. The two problems which comprise the project are intentionally posed in a slightly vague fashion to let you address them in a research mode, utilizing your CFD code as a tool of discovery.

9. *Please feel free to discuss this Project with me (Dan) or Ben Schaffer at any time*.

10. See note 9!

Project Tasks:

1. The first problem is posed primarily as a means to validate your code on a well-known flow: over a circular cylinder at low Reynolds number. Simulate the unsteady flow around the cylinder at $Re_D = 200$, with a constant surface temperature of 400K and a temperature of 300K at the inflow and upper and lower 'free lids.'



a.) Graphically illustrate the beginning and evolution of the Karman vortex street by displaying the vorticity, temperature and streamfunction at appropriate intervals.
b.) Determine the time-averaged Strouhal number of the shed vortices, and compare to the established value.
c.) Determine and plot the total heat transfer from the cylinder as a function of time: T(t) based on integrating $-k\frac{\partial T}{\partial n}$ where n is the normal to the surface.

2. Study a flow involving heat transfer that you find interesting.
   a.) This may involve an object with a complex geometry, for example
      i. A heat-exchanger application which might consist of an array of cylinders, or
      ii. A finned object, or
      iii. A simple geometry with a more complex thermal boundary condition (e.g. heat sources with radiation and convection)
   b.) Another approach may be to consider the possibility control, for example
      i. Passive control (e.g. surface heating/cooling to stabilize the flow over a body), or
      ii. Active control (e.g. activating a local heater/cooler in response to a sensor input to modify the flow/thermal environment in a desirable way).

   The notion of control will be discussed in class.

Finally… it is recognized that this project may represent one of your first forays into writing your own, rather substantial, simulation codes, and you may need help and guidance along the way, so, once again, do not hesitate at all to contact me (Dan) or Ben as questions or concerns arise!!