AOE/ME 4434/5434 (Adv) Introduction to CFD
Fall 2012
Instructor: Dr. Chris Roy

Semester Project
Due Wednesday, Dec. 12, 2012 at 8 pm ET

This is an individual project for those enrolled in the graduate course (AOE/ME 5434). If you are enrolled in the undergraduate course (AOE 4434), you have the option of working in two-person teams.

Write a CFD code to solve for the flow in a square lid-driven cavity with the top wall moving at velocity U_{lid}. Use the 2D incompressible Navier-Stokes equations with time derivative preconditioning (assuming constant temperature and viscosity) given by

$$\frac{1}{\beta^2}\frac{\partial p}{\partial t} + \rho\frac{\partial u}{\partial x} + \rho\frac{\partial v}{\partial y} = S$$
$$\rho\frac{\partial u}{\partial t} + \rho u\frac{\partial u}{\partial x} + \rho v\frac{\partial u}{\partial y} + \frac{\partial p}{\partial x} = \mu\frac{\partial^2 u}{\partial x^2} + \mu\frac{\partial^2 u}{\partial y^2}$$
$$\rho\frac{\partial v}{\partial t} + \rho u\frac{\partial v}{\partial x} + \rho v\frac{\partial v}{\partial y} + \frac{\partial p}{\partial y} = \mu\frac{\partial^2 v}{\partial x^2} + \mu\frac{\partial^2 v}{\partial y^2}$$

where S is an artificial viscosity term given by

$$S = -\frac{|\lambda_x|_{\max} C^{(4)}\Delta x^3}{\beta^2}\frac{\partial^4 p}{\partial x^4} - \frac{|\lambda_y|_{\max} C^{(4)}\Delta y^3}{\beta^2}\frac{\partial^4 p}{\partial y^4}$$

where $|\lambda_x|_{\max}$ is the magnitude of the largest eigenvalues in (x, t) space, $|\lambda_y|_{\max}$ is the magnitude of the largest eigenvalues in (y, t) space, and $C^{(4)}$ is a constant that generally lies in the range

$$\frac{1}{128} \le C^{(4)} \le \frac{1}{16}.$$

In the above equation, β is the time-derivative preconditioning term given by

$$\beta^2 = \max\left(u^2 + v^2, \kappa U_{lid}^2\right)$$

where κ can range from 0.001 to 0.9.

Use the simple explicit method (i.e., a point Jacobi method) with second-order accurate central differences to advance the discrete equations in pseudo-time until you reach the steady-state solution. For students enrolled in the graduate course (AOE/ME 5434), also implement an explicit (point) symmetric Gauss-Seidel scheme. Monitor iterative convergence using the steady-state iterative residuals (i.e., the steady portion of your discretization evaluated all at the same time level). A relative iterative convergence (i.e., the ratio of iterative residuals to initial iterative residuals at step 1) of *at least* 8 orders of magnitude is recommended. Use the stability criteria we discussed in class to determine the time step; you may choose local or global time stepping, but local time stepping is recommended (i.e., take the largest allowable time step at each node).

The Fortran, C, C^{++}, and MATLAB code templates you will be given have the capability to run manufactured solutions (by setting the *imms* input flag to one). Perform a code verification study by computing the discretization error norms for the manufactured solution case on a series of systematically-refined meshes. Use a Reynolds number of 10, compute the observed order of accuracy, and show that it approaches second-order with mesh refinement.

Results

Run baseline cases at Reynolds number of 100 with the following conditions:

$$\begin{aligned}
&\mathrm{Re} = \frac{\rho U_{lid} L}{\mu} = 100 \\
&0 \le x \le 0.05\, m \\
&0 \le y \le 0.05\, m \\
&L = 0.05\, m \\
&\rho = 1.0\, kg/m^3 \\
&U_{lid} = 1.0\, m/s
\end{aligned}$$

Examine the effects of mesh size, the κ parameter in the time derivative preconditioning, and the $C^{(4)}$ constant in the artificial viscosity. I recommend that the coarsest mesh you use be 65×65 nodes (starting with $i = j = 1$ at the lower left-hand corner where $x = y = 0$). The code is set up to perform pressure re-scaling to enforce a reference pressure at the center of the cavity, with the reference pressure p_{ref} = set to 0.801333844662 N/m^2(consistent with the manufactured solution). For those you taking the graduate course (AOE/ME 5434), also run Reynolds numbers of 500 and 1000 (note, these cases may require finer meshes to be stable). You will be supplied with numerical solutions from the literature (and possibly experimental data) for comparison purposes.

A formal project report is required which should include sections on the theory (governing equations and boundary conditions), discretization, stability, artificial viscosity, iterative convergence, code verification, discretization error estimation, and flow analysis. While the choice of programming languages is up to you, I will be giving you the code structure in Fortran 95/2003, C, C^{++}, and MATLAB and thus recommend that you write your program in one of these languages. You will be required to complete a number of subroutines as part of this project. Please note that if you choose to use MATLAB, it will likely run significantly slower than the Fortran/C/C^{++} codes. Include your CFD code as a separate file in your submission and also include it as an appendix in your report. Compare your code results to those that you obtain using a commercial or research CFD code. You will be given access to ANSYS/Fluent, which we will be discussing later in class. Limit your report (excluding appendices) to 20 pages. A report template will also be made available (although it is not required).

Put your code under version control using TortoiseSVN or similar version control software. Check your coding revisions into your repository frequently to ensure that you do not lose any of your work and to allow you to go back to previous versions of your code. Put a screen shot of a "diff" between two of your versions into an appendix of your report (use Alt-Print Screen or Function-Alt-Print Screen in Windows).

Note: The honor code will be strictly enforced on this project. Any individuals or teams caught cheating (e.g., using parts of someone else's program) will be reported to the Honor System.