Procedure For Reading The Couette Flow Datasets

Thomas S. Lund

NorthWest Research Associates, Colorado Research Associates Division
3380 Mitchell Lane, Boulder, CO 80301.

July 7, 2011

1 Overview

Each Couette flow data set is stored in a pair of of files. The first, with name of the form header.022500 is a small ascii file which contains the vital parameters. Here the extension 022500 is the time step at which the data was written. The actual flow data is stored in a much larger data file of the form vel.022500. The format of this file is ieee 64-bit little endian binary data. The file contains no formatting characters and can be read by just about any programming language. The supplied programs (discussed below) illustrate how to read the data with FORTRAN direct access read statements. Additional useful information can be found in the file input.dat which was used to specify the parameters when the flow solver was run. The information contained in input.dat is written to the header file, but the latter is not commented. It is most appropriate to have programs read the header file but humans will absorb the same information much more readily from the commented input data file.

The velocity file contains the fields u, v, w, [T], f_1, f_2, f_3, $C_p/2$, where u, v, and w are the velocity components in the x, y, z (streamwise, spanwise, wall-normal) directions respectively. T is the temperature, which is not included in incompressible runs with no buoyancy effects (no temperature data is included with the current data set). f_1, f_2, and f_3 are the mass fluxes, which will be discussed below. C_p is the pressure coefficient, defined as

$$C_p = \frac{p - p_0}{1/2\rho_0 U_0^2} \tag{1}$$

where U_0 is the wall velocity, p_0 is the mean hydrostatic pressure and ρ_0 is the constant density. The velocities and mass fluxes are normalized by U_0 and $\rho_o U_0$ respectively.

The mesh used to compute the Couette fields uses Nx+2, Ny+2, and Nz+2 points in the x, y, and z directions. Each direction contains two extra points, which serve to facilitate easy implementation of the boundary conditions. A sketch of the mesh in the x-z plane is shown in Figure 1.

To understand the mesh layout and boundary conditions, consider periodic boundary conditions applied in the x direction. The data of interest lies in the interval i=2,3,...,Nx,Nx+1. The boundary values are then set as follows u(i=1) = u(Nx+1) and u(Nx+2) = u(2). The

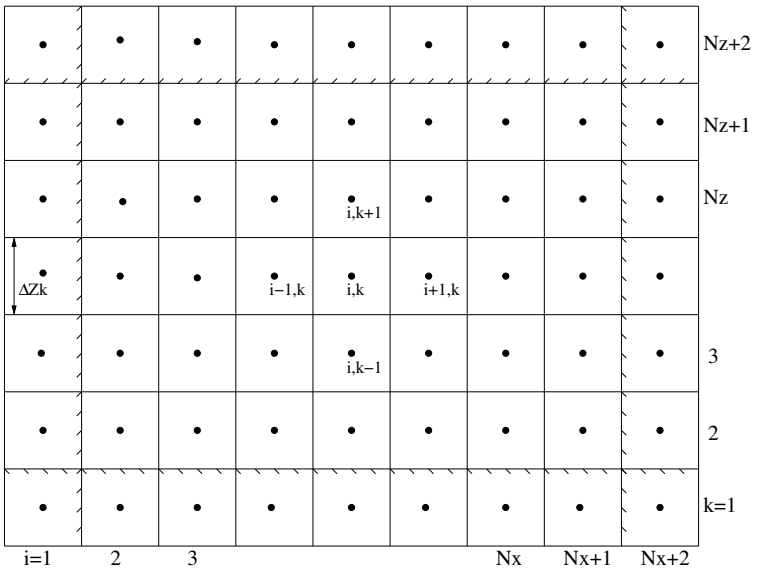

Figure 1: Computational mesh in the x-z plane. The extra rows of cells around the perimeter are used to facilitate a variety of boundary conditions.

advantage of this formulation is that finite difference operations can be taken naturally at the first interior point i=2, since valid data is available at i=1 and i=3.

The velocity, temperature and pressure are stored at the cell centers (denoted by dots in Figure 1). In order to balance fluxes in the finite-volume scheme, the mass fluxes across the various cell faces are also required. These are obtained by via interpolation from the cell-center values plus a divergence-free projection which insures mass conservation. The mass fluxes are not really needed for the present analysis, but they are nonetheless contained in the data file. It should also be mentioned that the cell-center velocity field is not exactly divergence free. This may cause a small wrinkle in working with the rate of strain tensor since it will not be exactly trace-free. As illustrated in the attached program, a remedy for this is to remove the residual trace in an isotropic fashion.

Using this mesh layout, the data in the velocity file is simply the array $u(i=1:Nx+2, j=1:Ny+2, k=1:Nz+2, n=1:n_{max})$, where n_{max} is 9 when temperature is included and 8 when it is not (8 for the cases here). The size of the data file in bytes is thus (Nx+2)*(Ny+2)*(Nz+2)*8*8, where the last factor of 8 is the conversion 8 bytes per 64-bit word.

2 Programs

A sample program which illustrates the procedure for reading the data is supplied in the compressed directory programs.zip. Download the zipped file and unzip it to create the directory programs. The directory will contain the three files read_couette.f, soda.h, and Makefile. The file soda.h is a header file which contains necessary information for the program read_couette. It is required but you do not need to understand what is in it. The file Makefile contains a few lines that illustrate how to compile the program. On a unix system

(or properly configured windows system) simply type make to compile the program. You may need to modify the first two lines of the makefile if you use a different compiler. In this case, make sure that you specify double precision (-r8 for the intel compiler) and bytes as the unit of measure for file record length (-assume byterecl for the intel compiler). Before running the program, download the header and vel file to the programs directory. Issue read_couette to run the program. When it prompts you for the file extension enter 022500. When the program finishes, you will see several new files with the extension 022500. These contain simple statistics or data written out along a vertical line. The names should be fairly self-explanatory.

3 Predicting Cavitation

Note that the pressure coefficient defined in Eq. (1) is very similar to the "cavitation number" defined in Buck's note on failure models for cavitation.

$$K = \frac{p - p_c}{1/2\rho_0 U_0^2} \tag{2}$$

where p_c is the pressure in the cavity. According to the cavitation number concept, cavitation would be predicted to occur anywhere that $K < K_{crit}$. By combining Eqs. (1) and (2) we can write

$$K = C_p + \underbrace{\frac{p_0 - p_c}{1/2\rho_0 U_0^2}}_{K_{shift}} \tag{3}$$

We can use the above formula to convert between the C_p data and the cavitation number. If we are going to use the cavitation number concept to predict cavitation, we can use the above equation to write a cavitation condition strictly in terms of the pressure coefficient, namely

$$C_p < K_{crit} + K_{shift} \tag{4}$$

As an example, let us assume that p_0 is 101325 Pascal (one atmosphere) and that the water used in the Couette flow experiment is at a temperature of 99.95 C, where the vapor pressure is 101146 Pascal. Assume further that $U_0 = 5$ m/s and that $\rho_0 = 1000$ kg/m^3, and that the cavity pressure p_c is equal to the vapor pressure. Finally, assume that the critical cavitation number is $K_{crit} = -0.030$. Equation (3) gives $K_{shift} = 0.0143$ and Eq. (4) then predicts that cavitation will occur if $C_p < -0.030 + 0.0143 = -0.0157$. For a given critical cavitation number we can adjust the water temperature and plate velocity in order to arrive at a reasonable distribution of cavitation sites.