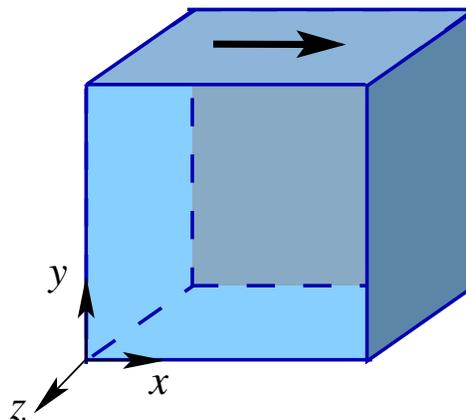


In this assignment you will compile and run the Fortran code contained in the folder **les18-stdnt** available at the course website: <http://courses.engr.uky.edu>.

This code is a large-eddy simulation program intended for studies of turbulence; but it can also be run for laminar flows as you will do in the problems of this assignment. You will investigate flows associated with the classical “lid-driven cavity” (LDC) problem, but in three space dimensions where much research still is ongoing. The basic geometry is displayed in the figure. Compilation will be done using the file **cmpl-les18**, and you will now need to use either the *FieldView* postprocessor, available for download only on the UK DLX supercomputer, or *TecPlot* available from Engineering Computing, for visualization of 3-D results.



The data file which must be read by the code is **inputfile.dat**, also available at the course website. You will make various modifications to this file to produce the range of data needed for the runs of this assignment. The basic task is to solve the 3-D Navier–Stokes (N.–S.) equations,

$$\mathbf{u}_t + \nabla \cdot (\mathbf{u}^2) = -\nabla p + \frac{1}{Re} \Delta \mathbf{u}, \quad \mathbf{x} \in (0, 1) \times (0, 1) \times (0, 1) \equiv \Omega, \quad t \in (0, t_f] \quad (1a)$$

with initial data

$$\mathbf{u}(\mathbf{x}, 0) \equiv 0, \quad \mathbf{x} \in \overset{\circ}{\Omega}, \quad (1b)$$

and boundary conditions

$$\mathbf{u}(\mathbf{x}, t) = 0, \quad \mathbf{x} \in \partial\Omega \setminus \{y=1 \forall x, z\}, \quad \text{and } u(x, 1, z, t) = 1, \quad v(x, 1, z, t) = w(x, 1, z, t) = 0. \quad (1c)$$

In these equations $\mathbf{u} \equiv (u, v, w)^T$; and $\mathbf{x} \equiv (x, y, z)^T$.

The divergence-free condition, $\nabla \cdot \mathbf{u} = 0$, is maintained by using the *projection 1* algorithm discussed in lecture, implemented on a 3-D staggered grid employing quasilinearization of nonlinear terms and centered differencing of all terms (except linearized advective terms appearing in the Jacobian matrix). Aliasing is treated with mollification via the Shuman filter, and associated filter parameters must, in general, be prescribed separately for each dependent variable. But, for this problem it is sufficient to assign the same value for all velocity components. The pressure Poisson equation is solved with successive line overrelaxation (SLOR), and the resulting pressure is not the true pressure but rather a potential used to project the velocity field onto a divergence-free subspace.

The purpose of this assignment is to provide some familiarity with behavior of N.–S. equation solutions as Re is increased and discretization step sizes are refined. In particular, you will investigate the influence of time step size, grid spacing and filter parameter values for three (3) different values of Re : 100, 1000, and 2000. In each of these cases you will employ (uniform) grid spacings of $h_x = h_y = h_z (= h)$ 0.1, 0.05, 0.025 with time steps corresponding to fixed

Courant number starting with $DT = 0.005$ on the 11^3 grid. You must investigate effects of filter parameter values for each Re , for each grid, and for each time step size. In general, you should always attempt to find the largest value of filter parameter β that leads to a stable (it doesn't blow up!) solution consistent with what is known experimentally for each value of Re . This requires numerical experimentation, which is one of the major disadvantages of this technique; but it offers so many advantages (in particular, simultaneous treatment of cell- Re and aliasing problems) that it is now commonly used—so, becoming accustomed to conducting the numerical experiments is useful.

To carry out the indicated studies you will begin with a very coarse 11^3 -point grid (too coarse for accuracy and, in fact, too coarse to even produce solutions at the higher values of Re with less robust codes). Be sure to run a sufficient number of time steps to attain steady solutions for the two lower values of Re (and a stationary solution for the highest value), and find nearly optimal values of β . You may wish to “play with” the various input parameters on this grid because executions are in general very fast. In particular, you may want to investigate β filter parameter values and values of the SOR parameter, ω , for the PPE solves. In general, lower values (~ 1.5) are appropriate for coarse grids, and higher ones—closer to 2.0—are better for fine grids. For low Re , filter parameter values can be expected to range from $\sim \mathcal{O}(10^2)$ – $\mathcal{O}(10^8)$ while for higher Re these will mainly be $\sim \mathcal{O}(10^2)$ – $\mathcal{O}(10^3)$; note that the values needed to maintain stability usually tend to decrease as Re increases. Moreover, you may wish to employ the restart capability on fixed grids as you increase Re : this will provide much better initial guesses than those given in Eq. (1b). This can be especially important on finer grids where much more time is required for each time step—and more time steps are needed to reach stationarity with constant Courant number.

Then you will repeat the calculations on finer grids. Be sure to record optimal values of β , as well as the SOR parameter ω , for each grid and for each Re . Moreover, record the final “physical” time needed for the solutions to reach steady state (or stationarity) at each Re , and, in particular, make plots of u, v, w vs. t using *gnuplot*, or whatever software you choose, to demonstrate these properties. These plots should be constructed at the center of the solution domain, and all three velocity components, as well as pressure, should be plotted for all three grids on the same graph for each solution component. Your final write up should contain a table in which all of these data are entered, namely, number of grid points and time step size, Re , β , ω and time to reach steady state, or stationarity, whichever is appropriate.

To complete this assignment, solve the following problems in a manner consistent with the above discussions.

1. **$Re = 100$.** First employ the $h = 0.1$ ($N_x = N_y = N_z = 11$) grid with time step size 0.005 and integrate to at least 25 seconds of physical time. Plot the time series, as discussed above, to verify that steady state has been achieved; if not, carry out longer integrations. Repeat for the two finer grids. Then plot time series at the center point of the cavity (e.g., $i, j, k = (6, 6, 6)$ for the 11^3 grid) for each of the four dependent variables; for each variable, plot results for all three grid spacings on the same plot (hence, four (4) plots) and assess the observed order of spatial accuracy.

At the final time step of the steady solution, plot u vs. y and v vs. x in the center z (x - y) plane. Then plot w vs. z in the center x - z plane at $y = 0.1$. Include results for all three grids on each plot. Also make contour plots of each velocity component and vorticity magnitude in the center x - y plane. (These four (4) plots should all be placed on the same

page for each combination of grid spacing and Re in your final report.) Note that the first of these plots (velocity profiles) can be enabled via *FieldView* or *TecPlot*, and these tools (or possibly *MatLab*) are essential for the contour plots.

Finally use your PLOT3D .xyz and final .qqq files in *FieldView* (or *TecPlot*) to construct 3-D streamlines, color coded with vorticity magnitude, with sufficient streamline density and cavity orientation to permit a good understanding of flow patterns. NOTE: you will need to employ the .nam file, **output3ldc.nam**, available at the course website, in order for *FieldView* to be able to build streamlines and calculate vorticity.

2. **$Re = 1000$** . Repeat Prob. 1 in detail, but note that filter parameter values may need to change. There is still a steady solution at this Re , as shown by both experiment and theoretical stability analyses.
3. **$Re = 2000$** . Now consider constructing only time-series plots for each of the three grid spacings. It is known from experiment (with some suggestion from stability analyses) that stationary flow behavior exists and should be quasiperiodic (at least two incommensurate—not related by rational numbers—frequencies); and although this can sometimes appear to be turbulent, it is not due to its lack of sensitivity to initial conditions. In constructing these plots (one for each solution component for each grid spacing) be sure to adjust the filter parameter value to achieve the behavior observed in experiments; and also be sure to have a sufficient period beyond beginning of stationarity to construct power spectra (say, using *MatLab*) if you wish—this is not required.

As in HW #1, with the results obtained by solving the above problems, prepare a journal-paper-like write up. Be sure to document the degree of grid-function convergence by qualitatively comparing results from the three different grids/time steps employed for each Re , but only for the optimal value of filter parameter. Thoroughly discuss these results, particularly the contour plots, in the context of the solution type (classical, etc.), and make recommendations as to whether the results would likely be useful, say, in a design context: can these results be trusted in a critical situation where human life is at stake? Suggest what might be done to improve the results based on your findings.