Parallel Simulation of Turbulent Flow in a 3-D Lid-Driven Cavity

J. M. McDonough
Departments of Mechanical Engineering and Mathematics
University of Kentucky, Lexington, KY USA 40506-0503
E-mail: jmmcd@uky.edu

Keywords: CFD benchmarks, turbulence, 3-D flows.

1. Introduction

The lid-driven cavity (LDC) problem has served as a benchmark for validation of CFD codes essentially from the beginning of CFD itself. The carefully-performed study of Burggraf (1966), although not the first investigation of this problem, provided early accurate 2-D results that still are cited today. Considerably later, with the advent of supercomputers, Ghia *et al.* (1982) published landmark computations of demonstrated accuracy that have long provided a quantitative benchmark against which to test CFD codes. Three-dimensional solutions to this problem began to appear by the mid to late 1980s, *e.g.*, Freitas *et al.* (1985), and work has continued to the present (for a review, see Shankar and Deshpande, 2005, and references therein). But only recently have studies of the calibur of the Ghia *et al.* 2-D analyses begun to appear (see Albensoeder and Kuhlmann, 2005). Clearly, the main impediment has been the same as it was in the early 2-D simulations—very long run times for 3-D problems. This highlights the importance of parallel computation and the utility of employing overall algorithms and numerical methods that are readily parallelizable.

Recent work on the 3-D LDC problem has been restricted to application of motion on at most top and bottom boundaries, and usually only on the top boundary. Although the resulting flow fields contain a large-scale vortical structure, essentially no matter what the value of Reynolds number, this structure does not necessarily lead to effective mixing. In the present study we introduce motion by sliding all planes of the cubical box containing the fluid and demonstrate in a qualitative way the more complicated streamlines compared with sliding a single plane. Of importance in the context of parallelization is analysis of parallel efficiency of the present 3-D code, which will ultimately be needed to enable thorough studies with guaranteed accuracy of results.

2. Analysis

Figure 1 displays the geometry of the problem considered here: a 3-D unit cube with unit velocities applied on each of the faces in opposite directions on opposing faces. It is apparent from simulations that this choice of velocity directions to some extent mitigates well-known pressure singularities in corners of the cube. The equations describing this flow situation are well known—the 3-D incompressible Navier–Stokes equations expressed here as

$$\nabla \cdot \boldsymbol{U} = 0 \tag{1a}$$

$$\frac{D\boldsymbol{U}}{Dt} = -\nabla P + \frac{1}{Re}\Delta \boldsymbol{U}\,, \tag{1b}$$

where $\boldsymbol{U} = (u, v, w)^T$ is the dimensionless velocity vector; P is pressure scaled with twice the dynamic pressure, and $Re \equiv UL/\nu$ with U and L being reference velocity and length, respectively (both set to un-

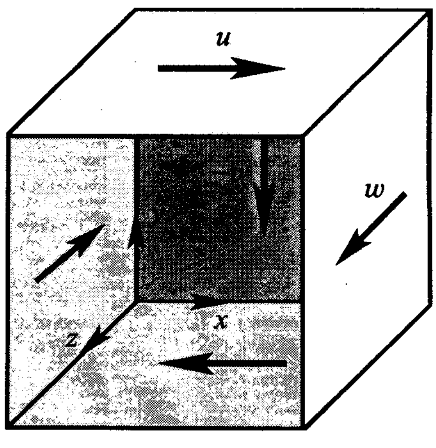

Figure 1. Basic problem geometry.

ity), and ν is kinematic viscosity. D/Dt is the usual substantial (material) derivative; ∇ is the gradient operator, and Δ is the Laplacian.

The code employed for this study has been reported previously (McDonough and Dong, 2001 and McDonough *et al.*, 2004) and consists of a new form of large-eddy simulation employing the following concepts: *i*) filter solutions rather than equations; *ii*) model subgrid-scale dependent variables, not their statistics; *iii*) employ subgrid-scale (SGS) results as part of a multi-scale formalism instead of discarding them. The numerical analysis is rather standard, but highly efficient and robust; it utilizes the following techniques implemented on a staggered, structured grid: centered differencing in space, trapezoidal integration in time with Douglas–Gunn (1964) time splitting, quasilinearization of nonlinear terms, (spatial) filtering of under-resolved large-scale results, all applied in the context of Gresho's (1990) projection 1 to handle velocity-pressure coupling. The code is parallelized via MPI as reported by McDonough and Yang (2005).

3. Results

In Fig. 2 we present some preliminary results for the LDC problem with shearing on all faces in the manner described above and indicated in Fig. 1. The Reynolds number employed for these calculations was 5000, and only 26^3 grid points were used in the simulations. Part (a) of the figure shows streamlines of complete (large-scale plus small-scale) velocity at a particular instant in time after a stationary (but not steady) behavior had been attained; part (b) displays small-scale streamlines at the same instant. In both cases streamlines are shaded with vorticity magnitude with dark blue corresponding to essentially zero, and pink indicating maximum, vorticity. We observe that orientations of the two parts of the figure are somewhat different; they correspond to views from the same general direction, thus permitting some qualitative comparison between small-scale and complete flow behaviors, but they are rotated somewhat differently to better view the small-scale motion. In particular, one can see a fairly large complicated vortical structure near the floor of the cavity in the complete flow field streamlines, and at the same time fairly intense small-scale turbulence activity in the same region.

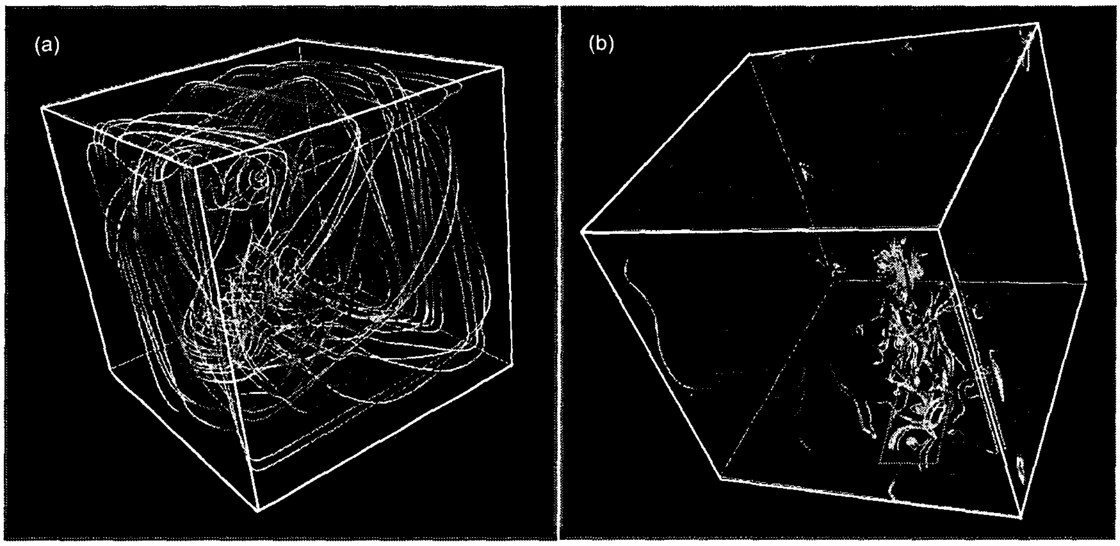

Figure 2. Velocity streamlines shaded with vorticity magnitude: (a) complete velocity; (b) small-scale velocity.

Figure 3 displays a zoom-in of the small-scale streamline field in the neighborhood indicated in Fig. 2(b) by the red box, but viewed from a somewhat different direction to highlight the size of flow structures produced by the form of LES employed in this study. We have very carefully aligned a copy of the computational grid so as to cross through the vortex that is relatively toward the front to show that the SGS turbulence model is able to produce structures significantly smaller than the resolved-grid scale. At the same time, it is clear from Fig. 2(b) that there are also both larger and smaller structures produced in the small-scale calculations thus at least suggesting a fairly good coverage of length scales and, correspondingly, an ability to produce direct interactions between large and small scales. Furthermore, as a consequence, there is no necessity to invoke a scale separation assumption when employing this type of model. We remark that this occurs predictably from the form of the SGS models. They rely on a sophisticated combination of deconvolution, generalization of Kolmogorov scaling and use of the "poor man's Navier–Stokes equations" (McDonough and Huang, 2004) to produce very physically-based small-scale behaviors.

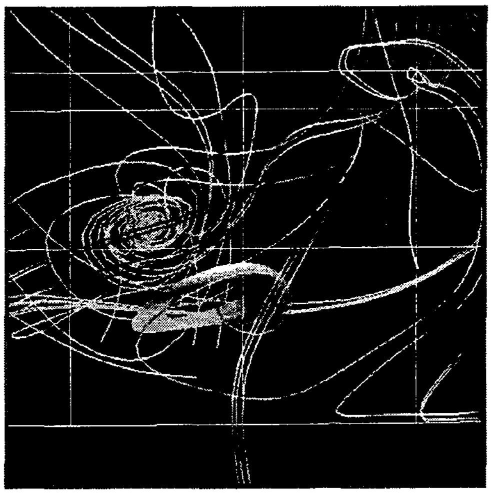

Figure 3. Small-scale streamlines zoom-in.

The problem discussed above has been run with three different grid spacings (26^3, 51^3 and 101^3) with a range of number of processors from two to 32. Results presented in Fig. 4 show speedups only for the finest of these grids. It should first be observed that the results are not especially good, and moreover these correspond to runs with completely dedicated processors. This is in significant contrast to earlier timings of parallelization with this code reported in the cited references. At present we have not yet determined the underlying cause(s) of this poor performance. There are, however, some specific differences between the present study and those reported earlier. First, the problem itself is somewhat different in that its geometry is uniform in all directions—there is no "longest" direction that in some instances can improve parallel performance. Second, there have been some changes to the code itself; but we did not expect these to affect parallelization. Finally, the processors now being used are different. The hardware is still a HP SuperDome SMP; but new processors were installed approximately six months ago, and while one would hope this would lead to improved performance we have not yet eliminated this as the cause for the poor parallel performance exhibited here.

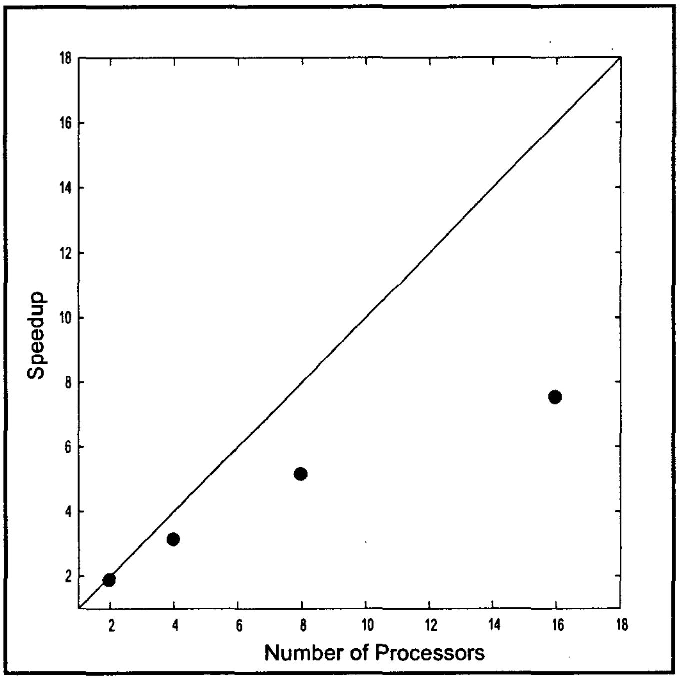

Figure 4. Speedup for a 101^3 gridpoint calculation.

4. Summary

We have introduced a new version of the 3-D lid-driven cavity problem that leads to more complicated fluid parcel trajectories and thus, enhanced mixing, but at the same time weakens corner singularities. We employed an advanced form of LES to solve this problem and presented preliminary results that show very complicated streamline structures on both large and small scales, despite a relatively low Reynolds number. Finally, we demonstrated moderate speedups via parallelization. Ongoing tests are expected to resolve the questions raised regarding possible sources of the rather poor parallel performance compared with that seen in earlier studies with the same code. Because it is expected that the findings may be significant for parallel performance in general, we plan to emphasize this aspect in the oral presentation at the *Parallel CFD 2006 Conference.*

References

S. Albensoeder and H. C. Kuhlmann (2005). Accurate three-dimensional lid-driven cavity flow, *J. Comput. Phys.* **206**, 536–558.

O. R. Burgraff (1966). Analytical and numerical studies of the structure of steady separated flows, *J. Fluid Mech.* **24**, 113–151.

J. Douglas, Jr. and J. E. Gunn (1964). A general formulation of alternating direction inplicit methods, part I. parabolic and hyperbolic problems, *Numer. Math.* **6**, 428–453.

C. J. Freitas, R. L. Street, A. N. Findikakis and J. R. Koseff (1985). Numerical simulation of three-dimensional flow in a cavity, *Int. J. Numer. Meth. Fluids* **5**, 561–575.

U. Ghia, K. T. Ghia and C. T. Shin (1982). High-*Re* solutions for incompressible flow using the Navier–Stokes equations and a multigrid method, *J. Comput. Phys.* **48**, 387–411.

P. M. Gresho (1990). On the theory of semi-implicit projection methods for viscous imcompressible flow and its implementation via a finite element method that also introduces a nearly consistent mass matrix. Part 1: Theory, *Int. J. Numer. Meth. Fluids* **11**, 587–620.

J. M. McDonough and S.-J. Dong (2001). 2-D to 3-D conversion for Navier–Stokes codes: parallelization issues, PARALLEL COMPUTATIONAL FLUID DYNAMICS *Trends and Applications*, Jenssen *et al.* (Eds.), North-Holland Elsevier Pub., Amsterdam, 173–180.

J. M. McDonough and M. T. Huang (2004). A 'poor man's Navier–Stokes equation': derivation and numerical experiments—the 2-D case, *Int. J. Numer. Meth. Fluids* **44**, 545–578.

J. M. McDonough and T. Yang (2005). Parallel performance of a new model for wildland fire spread predictions, PARALLEL COMPUTATIONAL FLUID DYNAMICS *Multidisciplinary and Applications*, Winter *et al.* (Eds.), Elsevier Pub., Amsterdam, 45–52.

J. M. McDonough, T. Yang and M. Sheetz (2004). Parallelization of a modern CFD incompressible turbulent flow code, PARALLEL COMPUTATIONAL FLUID DYNAMICS *Advanced Numerical Methods, Software and Applications*, Chetverushkin *et al.* (Eds.), Elsevier Pub., Amsterdam, 473–480.

P. N. Shankar and M. D. Deshpande (2000). Fluid mechanics in the driven cavity, *Annu. Rev. Fluid Mech.* **32**, 93–136.