

DIRECT NUMERICAL SIMULATION IN A LID-DRIVEN CAVITY AT HIGH REYNOLDS NUMBER

E. Leriche ^{†,*}

[†] Laboratoire d'Ingénierie Numérique, Institut des Sciences de l'Energie, Section de Génie Mécanique, Faculté des Sciences et Techniques de l'Ingénieur, Ecole Polytechnique Fédérale de Lausanne, Station 9, CH-1015 Switzerland. *Email: emmanuel.leriche@ep.ch

ABSTRACT

Direct numerical simulation of the flow in a lid-driven cubical cavity has been carried out at high Reynolds numbers (based on the maximum velocity on the lid), between $1.2 \cdot 10^4$ and $2.2 \cdot 10^4$. An efficient Chebyshev spectral method has been implemented for the solution of the incompressible Navier-Stokes equations in a cubical domain. The resolution used up to 5.0 million Chebyshev collocation nodes, which enable the detailed representation of all dynamically significant scales of motion. The mean and root-mean-square velocity statistics are briefly presented.

INTRODUCTION

Estimates for the attainable turbulent Reynolds number by the method of direct numerical simulation (DNS) have been known for several decades. This estimate is based on the ratio between the largest scales to the next ones (i.e. Kolmogorov scales), which scales like $Re^{3/4}$, where Re is the Reynolds number, and to resolve numerically all the scales, an upper bound in term of degrees of freedom (dof) is then given by $Re^{9/4}$. The evolution in computer hardware and algorithmic developments makes it now possible to extend the direct numerical simulation to transitional and turbulent flows that are inhomogeneous in all space directions. The present contribution is concerned with the numerical and physical aspects of the direct simulation of incompressible flow within a lid-driven cubical cavity. The fluid is enclosed in a cubical cavity where one wall is moved with a specified velocity. The goal is to reach the highest Reynolds number but in a very simple domain,

a cubical cavity and then to study in detail the three-dimensional transitional and turbulent flow properties within the cavity by means of direct simulation at high Reynolds numbers (based on the maximum velocity on the lid), between $1.2 \cdot 10^4$ and $2.2 \cdot 10^4$. The flow phenomena encountered within such systems are many and poorly understood.

THE GOVERNING EQUATIONS

The fluid enclosed in the cavity is assumed to be incompressible, viscous, Newtonian and homogeneous. The equation of motion for the fluid inside the cavity is given by the Navier-Stokes equations. The three-dimensional domain, denoted by Ω , is the open interval $(-h, +h)^3$ and its closure is written as $\bar{\Omega}$. The Navier-Stokes equations are written in vector notation as

$$\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla) \mathbf{u} = -\nabla p + \nu \Delta \mathbf{u} \quad (1)$$

with the continuity equation given by

$$\nabla \cdot \mathbf{u} = 0 \quad (2)$$

where the velocity vector \mathbf{u} has components $(u, v, w) \equiv (u_1, u_2, u_3)$, and $\mathbf{x} \equiv (x, y, z) \equiv (x_1, x_2, x_3)$.

The boundary conditions for the velocity consist in enforcing no-slip at every solid wall except on the moving lid at $y = +h$. The velocity distribution on the lid is given by the following expression

$$u(x, y = +h, z) = U_o(1 - (x/h)^n)^2(1 - (z/h)^n)^2$$

with $n = 18$ and where U_o is the maximum lid velocity. This lid velocity profile avoids singularity at the top edges of the cavity. The sensitivity of the results to polynomial distribution of the lid velocity profile is discussed in [4] for two-dimensional cavity flow. The Reynolds number is based on the maximum velocity on the lid and is given by $Re = U_o 2h/\nu$.

1 NUMERICAL APPROXIMATION

The spatial approximation of the equations of motion is based on the use of expansions in Chebyshev polynomials along every space direction. The collocation method consists of exactly enforcing the differential equations — the incompressible Navier-Stokes equations —, and the boundary conditions, at the Chebyshev-Gauss-Lobatto points [1]. A number of different methods are available to enforce the incompressibility constraint [9,2]. The Projection-Diffusion method analysed in [6,7] allows to decouple the velocity and pressure computation in very efficient way. This method proceeds by two steps, a pressure evaluation based on a Poisson equation with an extrapolated in time Neumann condition, and a time implicit diffusion step for the

nal velocity. The pressure is expanded into polynomials of the same order as the velocity, while intermediate velocity results are truncated in a way that enforces continuity without any contamination by spurious pressure modes. The projection-diffusion method is chosen for its consistency with the continuous space-time problem and for its optimal cost. The very simple geometry of the cavity allows to write a very efficient code based on solution of elliptic operators using the fast diagonalisation method [3]. The code is then highly vectorized and could reach more than 80 % of the peak performance of a single NEC processor (see Table 1). The time discretization of the Navier-Stokes equations is based on a second-order backward Euler differentiation formula. The viscous terms are treated implicitly whereas the non-linear terms are advanced explicitly in time by a second-order extrapolation scheme. They are evaluated from their skew-symmetric form. The pressure Neumann boundary conditions is found by a second-order extrapolation scheme. The overall accuracy is second-order in time and the scheme is subject to (CFL) restrictions in the time step due to the explicit advancement of the non-linear terms (see next section).

2 PHYSICAL AND COMPUTATIONAL PARAMETERS

The numerical simulations in the cubical cavity at Reynolds numbers of 12000 (resp. 18000 and 22000) have been performed on the NEC-SX4/5 at Swiss Center for Scientific Computing (CSCS-Manno) with a resolution of 129^3 (resp. 169^3) collocation points. The main computational and physical parameters are reported in Table 1. The time steps were chosen in order that the corresponding CFL number given by $\sum_{i=1}^3 \left| \frac{u_i \Delta t}{\Delta x_i} \right|$ are 10% below the CFL limit given in Table 1. Δx_i is the distance between two neighbouring collocation points in the x_i direction. The CFL_{MAX} turns

out to decrease when increasing the mesh size but also when increasing the Reynolds number. The spatial accuracy of the simulations is gauged by monitoring the evolution with time of the absolute values of the ratios of the lowest Chebyshev mode to that of the highest for each component of the velocity field in the spectral space. They did not exceed 10^{-3} . This is stricter criteria than those based on statistics such as spectra. The asymptotic behavior of the three velocity components near each boundary was also verified. An additional check was the balance of the terms in the equation for the turbulent kinetic energy –not shown here–. The databases are generated by storing the three-dimensional velocity and pressure fields in order to get first- and second-order statistics taken on meaningful sample. The sizes of those databases are provided in Table 1. It turns out that the statistics for the case of $Re=22000$ require much more longer sample than the one at $Re=18000$. The total integration time for the simulations reported in Table 1 leads to first- and second-order velocity statistics approximately symmetric about the mid-plane $z/h = 0$, the mid-plane being a plane of statistical symmetry. Both transient and steady-in-the-mean states of the flow possess long time scales requiring long integration times.

3 SOME DNS RESULTS

The most exhaustive quantitative experimental data for the lid driven cavity flow may be found in [8]. Measurements (mean and root-mean-square (*rms*)) on the vertical and horizontal centerlines in the mid-plane are reported at different Reynolds numbers up to 10000. The experimental data at the highest Reynolds number of 10000 provide a fair agreement with the simulation at the lowest Reynolds number of 12000 [5]. Nonetheless, this comparison is limited to the mid-plane. The first- and second-order velocity statistics have been computed

Domain size (x,y,z)	2h, 2h, 2h	2h, 2h, 2h	2h, 2h, 2h
$Re = U_o 2h/\nu$	12000	18000	22000
Nb. Chebyshev	129^3	169^3	169^3
Polynomials (x,y,z)			
Grid Resolution min	0.000301 h	0.000175 h	0.000175 h
Grid Resolution max	0.0245 h	0.0187 h	0.0187 h
Time step (h/U_o)	0.0025	0.0015	0.00125
CFL max	0.29	0.26	0.23
Computers (NEC)	SX4	SX5	SX5
Sustained G op rate (Peak)	1.8 (2.0)	6.5 (8.0)	6.5 (8.0)
Cpu(sec)/time step	20.625	17.00	17.00
Cpu(sec)/(time step*node)	$9.61 \cdot 10^{-6}$	$3.5 \cdot 10^{-6}$	$3.5 \cdot 10^{-6}$
Total integration time (h/U_o)	1000	1400	3500
Sampling rate (h/U_o)	0.25	0.1275	0.10625
Nb. of samples	4000	10000	30000
Size of one sample (Mb)	68.7	154.5	154.5
Total data held in store	250Gb	1.7Tb	5.4Tb

Table 1

Main physical and numerical parameters for direct numerical simulations at Reynolds numbers 12000, 18000 and 22000.

for the three Reynolds numbers. The effects of Reynolds number on the driven cavity flow will be preliminary discussed here. The kinetic energy K contained in the thin viscous layer of fluid close to the lid is successively transferred into the cavity flow by viscous diffusion. The total kinetic energy is shown to decrease with the Reynolds number (Fig. 1) in accordance with the estimate $K \approx U_o^2 (2h)^3 Re^{-1/2}$ given in [5]. The most dominant features of the mean flow

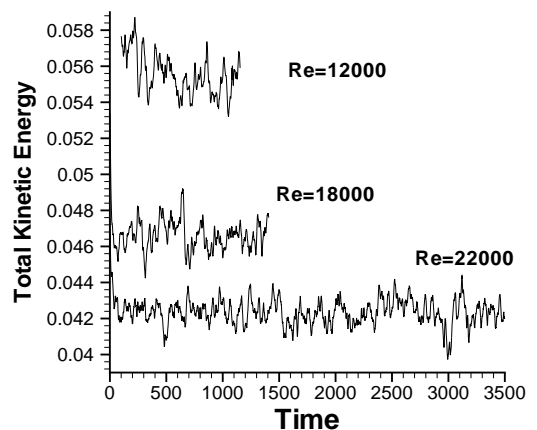


Fig. 1. Time history of the total kinetic energy for Reynolds numbers 12000, 18000 and 22000.

are the large-scale recirculation which spans the cavity, the coexistence of fast- and slow- flow

regions near the moving lid, the presence of wall jet (parallel to the downstream, bottom and, at higher Reynolds numbers, the upstream vertical wall), of jets impingement on the bottom and upstream wall, and of corner spiralling vortices [5]. Fig. 2 shows contours of the u and v mean fields in the mid-plane of the cavity. The viscous layer close to the lid and the downstream wall jet are getting thinner as the Reynolds increases. The enlargements of the downstream bottom corner clearly show that the size of corner vortices decrease as the Reynolds increases. Contours of the rms of u , v and w in the mid-plane near the downstream bottom corner are shown in Fig. 3. The rms increase with the Reynolds number, faster for u - rms and slower for v - rms . An analysis of the mean and fluctuating fields near the downstream bottom corner shows that the turbulence is generated by the impact of the flow descending near the downstream vertical wall (downstream wall jet). The jet impact on bottom wall can be easily determined by looking at the pressure distribution on the bottom wall –not shown here–. The peaks of pressure at the impact are getting narrower as the Reynolds increases. Along the downstream vertical wall, the flow is not turbulent but highly chaotic and near the bottom wall, the rms are found to reach their maximum values at the jets impact and in their vicinity, Fig. 3.

4 CONCLUSION AND FUTURE WORK

Direct numerical simulations using a Chebyshev collocation method have been performed for the lid-driven cavity flow at high Reynolds numbers (12000, 18000 and 22000) and first- and second order velocity statistics have been computed on long integration time to capture long time scales of the cavity flow. The mean momentum budgets are currently investigated and the leading terms in these balances are examined. The Reynolds stress budgets have been computed and the statistics

for the distribution of energy between the various components will be discussed.

BIBLIOGRAPHY

- [1] C. Canuto, M.Y. Hussaini, A. Quarteroni, and T.A. Zang. *Spectral Methods in Fluid Dynamics*. Springer Series in Computational Physics. Springer-Verlag, New-York, 1988.
- [2] M.O. Deville, P.F. Fischer, and E.H. Mund. *High-Order Method for Incompressible Fluid Flow*. Cambridge University Press, Cambridge, 2002.
- [3] P. Haldenwang, G. Labrosse, S.A. Abboudi, and M. Deville. Chebyshev 3D Spectral and 2D Pseudospectral Solvers for the Helmholtz Equation. *J. Comput. Phys.*, 55:115–128, 1984.
- [4] E. Leriche. *Direct Numerical Simulation of a Lid-Driven Cavity Flow by a Chebyshev Spectral Method*. PhD thesis, no:1932, Ecole Polytechnique Fédérale de Lausanne, Lausanne, 1999.
- [5] E. Leriche and S. Gavrilakis. Direct Numerical Simulation of the Flow in a Lid-Driven Cubical Cavity. *Phys. Fluids*, 12(6):1363–1376, 2000.
- [6] E. Leriche and G. Labrosse. High-order direct Stokes solvers with or without temporal splitting : Numerical investigations of their comparative properties. *SIAM J. Scient. Comput.*, 22(4):1386–1410, 2000.
- [7] E. Leriche, E. Perchat, G. Labrosse, and M.O. Deville. Numerical evaluation of the accuracy and stability properties of high-order direct Stokes solvers with or without temporal splitting. *Journal of Scientific Computing*, to appear, 2006.
- [8] A. K. Prasad and J. R. Koseff. Reynolds number and end-wall effects on a lid-driven cavity flow. *Phys. Fluids*, 1(2):208–218, 1989.
- [9] M. Schumack, W. Schultz, and Boyd J. Spectral method solution of the Stokes equations on nonstaggered grids. *J. Comput. Phys.*, 94:30–58, 1991.

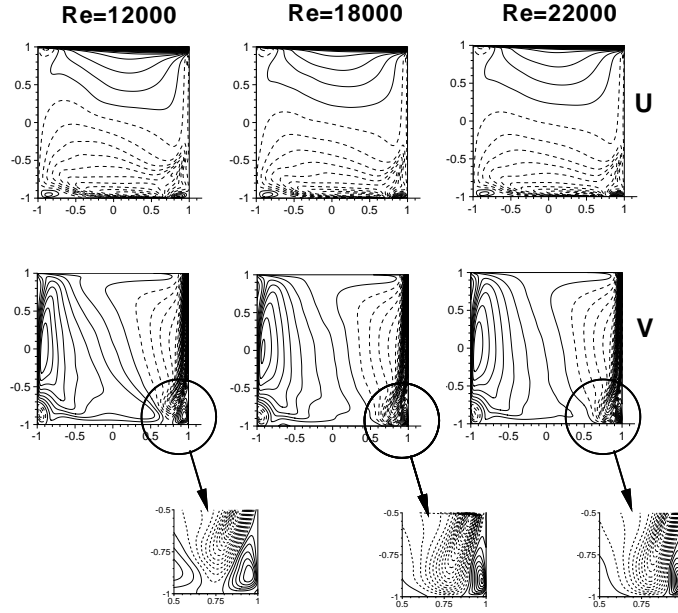


Fig. 2. Contours of the mean velocity field u and v in the mid-plane $z = 0$ (coordinates $(x/h, y/h)$). Top figures : equidistant mean velocity u contours; maximum contour level 1.0, minimum contour level -0.21, interval of 0.019. Bottom figures : equidistant mean velocity v contours; maximum contour level 0.1, minimum contour level -0.7, interval of 0.0125.

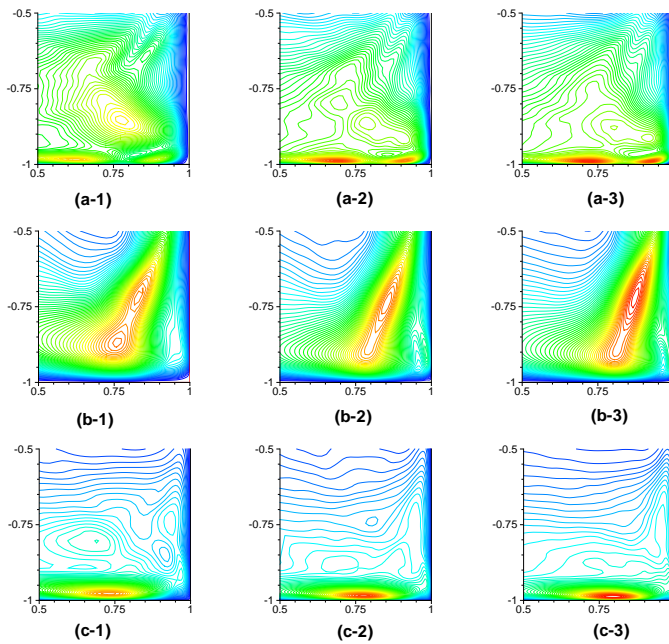


Fig. 3. Contours of the *rms* of u (a), v (b) and w (c) for the Reynolds numbers 12000 (1), 18000 (2) and 22000 (3) in downstream bottom corner of the mid-plane $z = 0$ (coordinates $(x/h, y/h)$). (a) equidistant u -*rms* contours; maximum contour level : 0.085, interval of 0.0013. (b) equidistant v -*rms* contours; maximum contour level : 0.16, interval of 0.0025. (c) equidistant w -*rms* contours; maximum contour level : 0.17, interval of 0.0027.