№. . . . .

ИНСТИТУТ ПО МАТЕМАТИКА С ИЗЧИСЛИТЕЛЕН ЦЕНТЪР

INSTITUTE OF MATHEMATICS WITH COMPUTER CENTER

A BLOCK - MATRIX ITERATIVE NUMERICAL METHOD FOR COUPLED SOLVING 2D NAVIER-STOKES EQUATIONS

Oleg P.Iliev,
Mikhail M.Makarov,

December 13, 1994

БЪЛГАРСКА
АКАДЕМИЯ
НА НАУКИТЕ

1869

BULGARIAN
ACADEMY
OF SCIENCES

PREPRINT

December 1994

No.10

Abstract

An algorithm for coupled solving 2D Navier - Stokes equations in the stream function ψ - vorticity ω variables is presented. Lid driven cavity flow is computed as a test example.Implicit difference schemes on uniform grids are used for discretizing the unsteady Navier - Stokes equations. An iterative method, similar to the BLOCK-ORTHOMIN(K) method, is used for solving block-matrix set of linear algebraic equations at each time step . The non-symmetric block is reversed on each block-iteration by using approximate factorization - ORTHOMIN(1) iterative method. The difference Laplace operator is reversed by means of a direct method. The comparison of the results, provided by coupled solving Navier-Stokes equations with those provided by decoupled (consecutive) solving the equations for ω and ψ, demonstrate the advantages of the suggested computing technique.

INTRODUCTION

Recently much interest has been devoted to the development of efficient algorithms for solving the system of Navier - Stokes equations because these equations are an important part of the mathematical modeling of various processes and phenomena. For a long time the decoupled solving numerical techniques have been mostly used for computing Navier - Stokes equations [1]. However the consecutive solving of the above equations leads to a restriction on the time step even when the implicit time approximation is used. Such a restriction may occur in the two most frequently used choices of variables: velocity u - pressure p formulation, and vorticity ω - stream function ψ formulation. In the first case a relation $u^{k+1} = f(p^k)$ arises. In the second case a relation

$$\omega^{k+1}\big|_\Gamma = f(\psi^k) \qquad (1)$$

has to be used for the consecutive solving the unsteady Navier - Stokes equations. Here k stands for the time level and Γ denotes the boundary of the domain under consideration. Lijumkis [2] and Vabishchevich [3] note that for the $\psi - \omega$ formulation and for moderate Reynolds numbers such as $Re < 1000$, the time-step restriction following from relation (1) is stronger then the time - step restriction caused by using the velocity values from the previous time step in the approximation of the convective terms. They have found numerically that the restriction when one uses (1) is:

$$\tau < 1.5\ Re\ h^2 \qquad (2)$$

where τ stands for the time - step, h stands for the mesh size. New difference schemes are proposed in [2,3] for relaxing the restriction (2).

Recently a number of papers has been devoted to coupled solving the system of Navier - Stokes equations, both, in $\psi - \omega$ and in $u - p$ formulations. Vanka in [4,5] proposes an algorithm for coupled solving Navier - Stokes equations in primitive variables using the finite difference method. He uses the multigrid technique in [6] for rapid computations. Rubin and Khosla [7] and Popov and Majorova [8] propose algorithms for coupled solving $\psi - \omega$ equations. Bender and Khosla [9] investigate the usage of direct sparse matrix solvers in the solving Navier - Stokes equations in $\psi - \omega$ formulation. Lipitakis [10] and Osswald et al. [11] use direct methods for coupled solving 3D Navier - Stokes equations. Arakawa et al. [12] compare results from the use of the multigrid technique for both, the coupled and the decoupled solving 2D Navier - Stokes equations. Van Dam and Hafez [13] compare some direct and iterative methods that have been used for solving particularly parabolized Navier - Stokes equations in $\psi - \omega$ formulation. Let us note that the direct methods have been used in many of the above papers for solving large sparse matrix equations at each time-step. However Radicati et al. [14] compare results from the use of iterative and direct methods for solving unsteady convection - diffusion equations and they show that for grids, finer than 64x32, the iterative methods are more preferable for computing the problem they consider. This conclusion is valid for the case when one solves the sets of linear algebraic equations with the same precision in both cases: using direct or iterative methods. However, Radicati et al. also note that in many cases there are no physical reasons for the very accurate solving of the matrix equation at each time step. Our own experience confirms these conclusions. From this point of view it is more preferable to use iterative methods to be able to solve the matrix equations at each time step with an appropriate accuracy. This conclusion is theoretically justified in a linear case: Bramble et al. [15] prove for a specific class of linear parabolic PDE's and for a certain class of numerical methods that it is sufficient to achieve a moderate accuracy for the numerical solution at each time step.

In the present paper a new iterative method for coupled solving 2D unsteady Navier - Stokes equations in $\psi - \omega$ formulation is proposed. The suggested numerical technique may be also used for computing steady-state problems, whose solution is considered as a time stabilization solution of the unsteady equations. The lid-driven cavity flow is computed as a test problem. Different approximations of the convective terms are used. The results from the coupled and the decoupled solving Navier - Stokes equations are compared for the above problem. It is demonstrated that the restriction (2) can be significantly relaxed for moderate Re (or, even removed for small Re) by the assistance of the suggested here coupled solving numerical technique. The presented numerical results illustrate the fact that the suggested technique allows the flow to be computed with the same accuracy (as in the case of using consecutive solving numerical technique) spending less CPU time.

The remainder of the paper is organized as follows. Next section is devoted to the mathematical model and the difference schemes. In third section the used block-matrix iterative method is described. In the last section the results from the numerical experiments are presented.

MATHEMATICAL MODEL AND DIFFERENCE SCHEMES

As it has been noted above, the lid-driven cavity flow is computed as a test problem. It is assumed that the lid of the cavity has suddenly started to move at the moment $t = 0$ with a dimensionless constant velocity $u = -1$. The unsteady dimensionless Navier - Stokes equations were computed up to the moment of fluid flow stabilization and the steady state solution is demonstrated for convenience.

The governing equations are:

$$\frac{\partial \omega}{\partial t} + \frac{\partial u\omega}{\partial x} + \frac{\partial v\omega}{\partial y} = \frac{1}{Re}\Delta\omega, \tag{3}$$

$$\Delta\psi = -\omega, \quad (x,y) \in \Omega, \quad t > 0. \tag{4}$$

Here $u = \frac{\partial \psi}{\partial y}$, $v = -\frac{\partial \psi}{\partial x}$ are components of the velocity vector, ψ is the stream function and $\omega = \frac{\partial v}{\partial x} - \frac{\partial u}{\partial y}$ is the vorticity of the velocity. Ω denotes the unit square in R^2, Γ is its boundary, Re stands for Reynolds number. The usual non-slip and impermeability boundary conditions are considered:

$$\psi = 0, \quad \frac{\partial \psi}{\partial n} = 0, \quad (x,y) \in \Gamma/\Gamma^1, \tag{5}$$

$$\psi = 0, \quad u = \frac{\partial \psi}{\partial y} = -1, \quad (x,y) \in \Gamma^1 = \{(x,y):\ 0 < x < 1,\ y = 1\}. \tag{6}$$

An uniform rectangular grid $\bar{\Omega}_h = \Omega_h \cup \gamma_h$ with steps h_x and h_y, and sizes N_x and N_y, is introduced on $\Omega \cup \Gamma$. Implicit finite difference schemes are used to approximate the system (1)-(4) and in the general case they may be written as follows:

$$\begin{bmatrix} A_{11} & A_{21} \\ A_{12} & A_{22} \end{bmatrix} \begin{bmatrix} \hat{\omega} \\ \hat{\psi} \end{bmatrix} = \begin{bmatrix} f_1 \\ f_2 \end{bmatrix} \tag{7}$$

The blocks and functions in (7) may be written in the more detailed form as follows:

$$A_{11}\hat{\omega} = \begin{cases} I\hat{\omega} + \tau A_c(\psi, \hat{\omega}) - \frac{\tau}{Re}\Lambda\hat{\omega}, & (x,y) \in \Omega_h, \\ I\hat{\omega} & (x,y) \in \gamma_h, \end{cases} \tag{8}$$

$$A_{12}\hat{\psi} = \begin{cases} 0, & (x,y) \in \Omega_h, \\ -\frac{2}{h^2}\hat{\psi}_{s-1}, & (x,y) \in \gamma_h/\gamma_h^1, \\ -\frac{2}{h^2}\hat{\psi}_{s-1} + \frac{2}{h}, & (x,y) \in \gamma_h^1, \end{cases} \tag{9}$$

where s-1 denotes the nearest node on the internal boundary normal,

$$A_{21}\hat{\omega} = \begin{cases} I\hat{\omega}, & (x,y) \in \Omega_h, \\ 0, & (x,y) \in \gamma_h, \end{cases} \tag{10}$$

$$A_{22}\hat{\psi} = \begin{cases} \Lambda\hat{\psi}, & (x,y) \in \Omega_h, \\ I\hat{\psi}, & (x,y) \in \gamma_h, \end{cases} \tag{11}$$

$$f_1 = \begin{cases} I\omega, & (x,y) \in \Omega_h, \\ 0, & (x,y) \in \gamma_h, \end{cases} \tag{12}$$

$$f_2 = \begin{cases} 0, & (x,y) \in \Omega_h, \\ 0, & (x,y) \in \gamma_h, \end{cases} \tag{13}$$

It can be seen that in this case A_{11}, A_{12}, A_{21}, f_1 and f_2 are the same as for the usually used decoupled numerical techniques [1]. Let us note that the operator A_{12} includes Thom' boundary conditions for the vorticity, but opposite to (1) in this case we have

$$\omega^{k+1}\big|_{\Gamma} = f(\psi^{k+1}) \tag{14}$$

The grid functions in (8)-(13) are denoted by the same letters as the continuous functions and the following notations are used: $\hat{\omega} = \omega(x, y, t^{k+1})$, $\omega = \omega(x, y, t^k)$. I is the identity operator, $A_c(\psi, \hat{\omega})$ is a linear grid operator approximating the convective terms, Λ is a grid operator approximating 2D Laplace operator on the uniform grid. The different choices of the operator A_c determine the different difference schemes:

CD: $A_c(\psi, \hat{\omega})$ -the central differencing of the convective terms;
FUD: $A_c(\psi, \hat{\omega})$ -the first upwind differencing scheme [1];
SUD: $A_c(\psi, \hat{\omega})$ -the second upwind differencing scheme [7].

Using the consecutive solving algorithm we will use notations CS.CD, CS.FUD, CS.SUD, respectively.

ITERATIVE METHOD

Let us rewrite the set of linear algebraic equations (7) as follows:

$$Aw = g \tag{15}$$

where

$$A = \begin{bmatrix} A_{11} & A_{21} \\ A_{12} & A_{22} \end{bmatrix}, \quad w = \begin{bmatrix} w^1 \\ w^2 \end{bmatrix}, \quad g = \begin{bmatrix} g^1 \\ g^2 \end{bmatrix},$$

$g^1, g^2, w^1, w^2 \in R^N$; $A_{11}, A_{12}, A_{21}, A_{22} \in R^{N\times N}, N = N_x \times N_y$. The matrix A and submatrices A_{11}, A_{22} are assumed to be nonsingular. The following algorithm describes a preconditioned iterative method BLOCK-ORTHOMIN(K) [16] for solving (15):

(i) solve $B\xi^{(n)} = g - Ay^{(n)}, n = 0, 1, \ldots;$

(ii) compute $p = \begin{cases} \xi^{(n)}, & n = 0; \\ \xi^{(n)} - \sum_{l=1}^{min(K,n)} b_{n,n-l} p^{n-l}, & n = 1, 2, \ldots \end{cases}$

$$b_{n,n-l} = \frac{(D\xi^{(n)}, p^{(n-l)})}{(Dp^{(n-l)}, p^{(n-l)})}, \quad l = 1, 2, \ldots min(K, n);$$

(iii) compute $y^{(n+1)} = y^{(n)} + a_n p^{(n)}, \quad a_n = \frac{(Dz^{(n)}, p^{(n)})}{(Dp^{(n)}, p^{(n)})},$

where $z^{(n)} = w - y^{(n)}, n = 0, 1, \ldots;$ B is the preconditioner and the choice of D, symmetric and positive definite, allows to compute inner product $(Dz^{(n)}, p^{(n)})$ for every n. For our case of block-matrix A, assuming that the equations

$$A_{11} x^1 = g^1 \tag{16}$$

and

$$A_{22} x^2 = g^2 \tag{17}$$

can be easily solved, the following choice of B and D may be tried:

$$B = (2I - AD_A^{-1})^{-1} D_A,$$

$$D = (D_A^{-1} A)^T (D_A^{-1} A).$$

Here I is the identity matrix and

$$D_A = \begin{bmatrix} A_{11} & 0 \\ 0 & A_{22} \end{bmatrix}.$$

For any special initial guess such that

$$r^{(0)} = g - Ay^{(0)} = \begin{bmatrix} r^1 \\ 0 \end{bmatrix}, \tag{18}$$

only one equation (16) and one equation (17) have to be solved at each iteration step of the BLOCK-ORTHOMIN(K) method. Generally, this is not true in the case of the iterative solving of (16) or (17) with insufficient accuracy, so restarts may be recommended updating the iterative guess to satisfy the condition (18).

In our case method of approximate factorization MAF by Dupont et al. [17], modified and coded by Makarov [18], is used for solving equation (16). March algorithm by Bank and Rose [19], modified and coded by Kaporin [20], is used for solving equation (17).

NUMERICAL RESULTS AND CONCLUSIONS

The lid-driven cavity flow is computed on uniform grids with 33x34 and 65x66 nodes. Values of the Reynolds number equal to 10, 100, 400, 1000 are considered. For comparison, the lid-driven cavity flow is computed by use of the coupled solving numerical technique (7)-(13), as well as by use of the usual decoupled (consecutive) solving technique [1]. In both cases the same methods and codes are used for solving equations (16) and (17). The following criterion of flow stabilization is used:

$$\|(\hat{\omega} - \omega)/\tau\|_{L_2} < \varepsilon \|\hat{\omega}\|_C$$

All computational results are presented in Tables I-V. The following computed data are presented in Tables I-IV for Reynolds number equal to 10,100,400, and 1000, respectively:

- CPU: CPU time in seconds on main frame computer IBM 4341;
- NSTEP: number of time steps up to flow stabilization;
- NBIT: number of block-iterations throughout all time steps;
- NALLIT: overall number of iterations for solving systems (16) throughout all time steps;
- ψ_{max}: maximum value of stream function.

As it was mentioned above, the schemes used in our computations are denoted as follows: CD - central differencing scheme, FUD - first upwind differencing scheme [1], SUD - second upwind differencing scheme [7]. In the case of using the consecutive solving numerical technique, the schemes are denoted CS.CD, CS.FUD, and CS.SUD, respectively. For $Re = 1000$ only upwind schemes are used.

It can be seen from Tables I-IV that restriction (2) does not take place when one uses the coupled solving numerical technique. For small Reynolds numbers (10 and 100) any value of the time step t can be chosen for any of considered difference schemes. For moderate Reynolds numbers (400 and 1000) the same is true only when one uses upwind schemes. This possibility very large time steps to be used for solving unsteady Navier-Stokes equations shows that the implicitness of the boundary conditions for the vorticity is more important than the nonlinearity of the convective terms for lid-driven cavity flow for $Re < 1000$. Note that if the value of t is very large we have to consider the Navier - Stokes equations solution as a solution of the steady state problem obtained by a simple iteration method. In this case t is not the time step but the iteration parameter. The presented in Tables I-IV results also show that for small and moderate values of the Reynolds numbers there are no reasons for using Newton's method for linearization of the considered equations.

Note, that the steady state lid driven cavity flow can be computed faster with the coupled solving numerical technique, as it can be seen from Tables I-IV for fixed Re number and for chosen difference scheme (the time step values for the consecutive solving technique are chosen in agreement with the investigations from our paper [23]).

The main reason for this is that the coupled solving numerical technique allows larger values for the time step t to be used. Note that suggested computing technique (7)-(13) is essentially more effective for small and moderate Reynolds numbers and on fine grids (see Tables I and II). For $Re = 1000$ the both, coupled and decoupled, numerical techniques require relatively the same computational resources. Table V presents more detailed information about computed flows for $Re = 100, 400$, and 1000, as well as data from other papers. One can observe there the stream function value and the vorticity value at the primary vortex center (4^{th} and 5^{th} columns), at the left corner vortex center (6^{th} and 7^{th} columns), and at the right corner vortex center (8^{th} and 9^{th} columns), respectively. The last column presents the vorticity value at the mid of the lid. The results, computed by using the coupled solving numerical technique, as well as by using the decoupled (consecutive) technique, are presented. For comparison, the corresponding data from papers of Chia et al.[21], Gupta [22], and Vanka [6] are presented. In general, there exist a good agreement between data, computed here and data, computed by other authors. The three different spatial approximation of the convective terms used in this paper are wellknown from a long time. Therefore we will not discuss in details their advantages and disadvantages. We will just briefly discuss the results presented in the Tables I-V, concerning the accuracy of the computations. As one may expect, all the three used difference schemes are very similar for small Reynolds numbers and they produce almost identical results in this case (see Table I). It can be seen from Table V that the larger value for the Reynolds number is considered, the less adequate results are obtained using the first upwind differencing scheme. Concerning the central differencing (CD) and second upwind differencing scheme (the last one has almost second order spatial approximation in the regions of the slow flow). Both these schemes give almost the identical values of the stream function and of the vorticity at the primary and secondary vortices centers. These values are in good agreement with the respective values from papers [6,21,22] (note, that we use a grid with 65x66 nodes, Chia et al.[21] use 129x129 nodes, Vanka [6] uses 321x321 nodes, and Gupta [22] uses 44x44 nodes, but he uses higher order difference schemes).

Summarizing, the explicit determination of the boundary conditions for the vorticity (1) is the main restriction on the time step in solving 2D unsteady Navier - Stokes equations in stream function - vorticity formulation when the closed domain and the moderate Reynolds numbers are considered. The suggested above computing technique allows the restriction (2) to be overcome.

Acknowledgments

We wish to thank Alexander Churbanov, Inst. for Math. Modeling, Russian Acad. Sci., for providing us with an extensive bibliography and for helpful comments on our work. The first author was partially supported by Bulgarian fund for scientific research under grant No. MM-415.

REFERENCES

1. P.J.Roache, Computational Fluid Dynamics, Hermosa Publishers, 1976.
2. E.D.Lijmkis, 'On the increasing time-steps when solving Navier - Stokes equations in the vortex - stream function formulation', Differentsial'nye uravnenia, 21,No.7,1208-1217(1985).(Russian).
3. P.N.Vabishchevich, 'Implicit difference schemes for non - stationary Navier - Stokes equations stream function - vorticity variables, Differentsial'nye uravnenia,20,No.7,1135-1144(1984)(English translation: Differential Equations, 20, No.7, 820-827(1984)).
4. S.P.Vanka and G.K.Leaf, 'An efficient finite - difference calculation procedure for multi - dimensional fluid flows', AIAA - 84, pap.1244.
5. S.P.Vanka, 'Block-implicit calculation of steady turbulent recirculating flows', Int.J. Heat Mass Transfer, 28, No.11, 2093 - 2103(1985).
6. S.P.Vanka, 'Block-implicit multigrid solution of Navier-Stokes equations in primitive variables', J. Comp. Phys., 65, 138-158(1986).
7. S.G.Rubin and P.K.Khosla, 'Navier - Stokes calculation with a coupled strongly implicit method', Computer and Fluids, 9,163-180 (1981).
8. O.S.Majorova and Yu.P.Popov, 'Numerical methods for solving Navier - Stokes equations', Soviet J. Comp. Math. Math. Phys., 20, No.4, 1005-1020(1980).
9. E.E.Bender and P.K.Khosla, 'Solution of the 2D Navier - Stokes equations using sparce matrix solvers' AIAA-87, pap.603.
10. E.A.Lipitakis, 'A sparse linear equations solver for 3d regular domain' Comm. Appl. Num. Meth., 3,201-213(1987).
11. G.A.Osswald, K.N.Chia and U.Chia, 'A direct algorithm for solution of incompressible 3D unsteady Navier-Stokes equations' AIAA. 8 Comp. Fluid Dyn. Conf.,1987.
12. C.Arakawa, A.O.Demuren, W.Rodi and B.Shoenung, 'Application of MULTI-GRID methods for the coupled and decoupled solution of the incompressible Navier-Stokes equations', Proc.7 GAMM Conf. on Num. Meth. in Fluid Mechanics, Lonvian-la Nenve, Lelgium. Sept.9-11, 1987. Brunswick,FRG,Friedr.Viewg und Sohn,1988,p.1-8.
13. C.P. van Dam and M.Hafez,'Comparison of iterative and direct solution methods for viscous flow problems', AIAA J.,27, No.10, 1459-1461(1989).
14. J.H.Bramble, J.E.Pasiak, P.H.Sammon and V.Thomee, 'Incomplete iterations in multistep backward difference methods for parabolic problems with smooth and nonsmooth data', Math. Comp., 52, No.186, 339-367(1989).

15. G.Radicatti, Y.Robert and S.Succi, 'Iterative algorithms for solution of non-symmetric systems in modeling of weak plasma turbulence', J.Comp. Phys.,80, No.2,489-498(1989).

16. L.A.Hageman and D.M.Young, Applied Iterative Methods, Academic Press, New York,1981.

17. T.Dupont, R.P.Kendel and H.H.Rachford, 'An approximate factorization procedure for solving self-joint elliptic difference equations', SIAM J. Numer. Anal., 5,559-573(1968).

18. M.M.Makarov, In: Num. Meth. for Solving Mathematical Physics Problems, Moscow State University, Moscow, p.174(1986). (Published by VINITY, in Russian).

19. R.E.Bank and D.J.Rose,'Marching algorithms for elliptic BVPs, part I. The constant coefficient case', SIAM J.Numer.Anal.14,792-829 (1977).

20. I.E.Kaporin, 'Modified marching algorithm for solving difference equations approximating Dirichlet problem for Poisson equation in a rectangular', In: Difference Methods for Mathemaical Physics Problems, (Yu.P.Popov and E.S.Nikolaev, eds.) 1980, pp.11-21(Russian) .

21. U.Chia, K.N.Chia, and C.T.Shin, 'High-Re solutions for solving incompressible flow using the Navier - Stokes equations and a multigrid method', *J. Comp. Phys.*, 48, 387-411(1982).

22. M.M.Gupta, 'High accuracy solutions of incompressible Navier - Stokes quations', *J. Comp. Phys.*, 93, 343-359(1991).

23. O.P.Iliev, M.M.Makarov, and P.S.Vassilevski, 'Performance of certain iterative methods in solving implicit difference schemes for 2D Navier - Stokes equations', *Int. J. Num. Meth. Engng.*, 33, 1165 - 1479(1992).

Table 1. $Re = 10$.

mesh size	τ	SCHEME	CPU	NSTEP	NBIT	NALLIT	ψ_{max}
33x 34	0.4	CD	42	10	25	240	0.09972
	4.0	CD	23	6	13	128	0.09970
	0.01	CS.CD	80	132		253	0.09972
	0.4	FUD	42	10	25	241	0.09980
	4.0	FUD	24	6	14	136	0.09983
	0.01	CS.FUD	86	132		261	0.09990
	0.4	SUD	45	10	25	264	0.09973
	4.0	SUD	25	6	14	136	0.09976
	0.01	CS.SUD	94	132		258	0.09971
65x 66	0.1	CD	480	16	86	763	0.09998
	10.0	CD	163	4	28	280	0.10000
	0.003	CS.CD	854	389			0.09998
	0.1	FUD	541	17	100	856	0.09996
	10.0	FUD	154	4	28	253	0.09998
	0.003	CS.FUD	924	388		596	0.1001
	0.1	SUD	598	17	106	971	0.09999
	10.0	SUD	164	4	28	278	0.10010
	0.003	CS.SUD	1005	389		593	0.09998

Table II. $Re = 100$.

mesh size	τ	SCHEME	CPU	NSTEP	NBIT	NALLIT	ψ_{max}
33x34	1.0	CD	60	16	39	344	0.1021
	10.0	CD	69	15	41	450	0.1022
	0.1	CS.CD	91	138		395	0.1021
	1.0	FUD	80	22	53	456	0.09982
	10.0	FUD	33	9	21	188	0.09980
	100.0	FUD	26	7	17	148	0.09965
	0.1	CS.FUD	94	127		392	0.1014
	1.0	SUD	78	20	46	451	0.1021
	10.0	SUD	44	9	26	256	0.1022
	100.0	SUD	27	7	17	154	0.1021
	0.1	CS.SUD	91	111		383	0.1021
65x66	1.0	CD	285	19	51	422	0.1032
	10.0	CD	374	13	68	586	0.1027
	0.03	CS.CD	997	410		866	0.1031
	1.0	FUD	431	20	73	659	0.09994
	10.0	FUD	158	9	25	238	0.09997
	100.0	FUD	100	6	16	152	0.09969
	0.03	CS.FUD	1210	469		994	0.1029
	1.0	SUD	385	20	62	576	0.1032
	10.0	SUD	191	10	30	305	0.1034
	100.0	SUD	460	8	32	1045	0.1040
	0.03	CS.SUD	1147	410		898	0.1031

Table III. $Re = 100$.

mesh size	τ	SCHEME	CPU	NSTEP	NBIT	NALLIT	ψ_{max}
65x 66	1.0	CD	780	40	110	1313	0.1122
	0.1	CS.CD	867	302		1006	0.1120
	1.0	FUD	603	43	99	864	0.09978
	10.0	FUD	183	10	30	280	0.09968
	0.1	CS.FUD	808	271		879	0.1037
	1.0	SUD	636	40	99	952	0.1122
	10.0	SUD	300	13	51	460	0.1122
	0.1	CS.SUD	914	299		694	0.1120

Table IV. $Re = 1000$.

mesh size	τ	SCHEME	CPU	NSTEP	NBIT	NALLIT	ψ_{max}
33x 34	10.0	FUD	42	12	24	232	0.09941
	100.0	FUD	27	7	15	148	0.09969
	1.0	CS.FUD	52	51		322	0.07771
	10.0	SUD	160	29	101	936	0.1030
	100.0	SUD	61	11	35	377	0.1037
	0.25	CS.SUD	144	184		529	0.1025
65x 66	5.0	FUD	410	28	64	614	0.09944
	100.0	FUD	104	6	15	144	0.09953
	0.25	CS.FUD	508	148		704	0.09464
	5.0	SUD	425	31	62	620	0.1162
	100.0	SUD	594	14	74	1149	0.1126
	0.25	CS.SUD	581	166		718	0.1141

Table V.

Re	mesh size	SCHEME	primary vortex ψ_{vc}	primary vortex ω_{vc}	left corner vortex ψ_{lcvc}	left corner vortex ω_{lcvc}	right corner vortex ψ_{rcvc}	right corner vortex ω_{rcvc}	$\omega(.5,1)$
100	65	FUD	0.09994	3.11	-0.297(-5)	-0.016	-0.296(-5)	-0.016	5.87
		CS.FUD	0.1029	3.10	-0.253(-5)	-0.021	-0.114(-4)	-0.030	6.65
		SUD	0.1032	3.11	-0.144(-4)	-0.040	-0.253(-5)	-0.020	6.61
		CS.SUD	0.1031	3.11	-0.145(-4)	-0.040	-0.250(-5)	-0.021	6.61
		CD	0.1031	3.11	-0.146(-4)	-0.040	-0.252(-5)	-0.020	6.61
		CS.CD	0.1031	3.11	-0.145(-4)	-0.040	-0.250(-5)	-0.021	6.61
	129	Chia[21]	*0.1034*	*3.17*	*-0.125(-4)*	*-0.031*	*-0.175(-5)*	*-0.016*	
	44	Gupta[22]	*0.1032*	*3.28*	*-0.124(-4)*		*-0.174(-5)*		*6.56*
	321	Vanka[6]	*0.1034*		*-0.114(-4)*		*-0.194(-5)*		
400	65	FUD	0.09978	3.26	-0.303(-5)	-0.015	-0.303(-5)	-0.015	5.88
		CS.FUD	0.1037	2.19	-0.308(-3)	-0.251	-0.997(-5)	-0.047	10.13
		SUD	0.1122	2.27	-0.676(-3)	-0.418	-0.167(-4)	-0.049	10.36
		CS.SUD	0.1120	2.26	-0.673(-3)	-0.417	-0.165(-4)	-0.049	10.38
		CD	0.1122	2.27	-0.676(-3)	-0.418	-0.167(-4)	-0.049	10.36
		CS.CD	0.1120	2.26	-0.673(-3)	-0.417	-0.165(-4)	-0.049	10.38
	129	Chia[21]	*0.1139*	*2.29*	*-0.642(-3)*	*-0.434*	*-0.142(-4)*	*-0.057*	
	44	Gupta[22]	*0.1112*	*2.30*	*-0.700(-3)*		*-0.137(-4)*		*10.15*
	321	Vanka[6]	*0.1136*		*-0.645(-3)*		*-0.146(-4)*		
1000	65	FUD	0.09920	3.24	0.310(-5)	-0.015	-0.311(-5)	-0.015	5.91
		CS.FUD	0.09464	1.74	0.658(-4)	-0.151	-0.839(-4)	-0.668	16.90
		SUD	0.1141	2.01	-0.228(-3)	-0.297	-0.190(-2)	-1.100	16.13
		CS.SUD	0.1141	2.01	-0.226(-3)	-0.296	-0.190(-2)	-1.100	16.14
	129	Chia[21]	*0.1179*	*2.05*	*-0.231(-3)*	*-0.362*	*-0.175(-2)*	*-1.155*	
	44	Gupta[22]	*0.1074*	*2.01*	*-0.138(-3)*		*-0.211(-2)*		*16.24*
	321	Vanka[6]	*0.1173*		*-0.224(-3)*		*-0.174(-2)*		