Possibility of Implicit LES for Two-Dimensional Incompressible Lid-Driven Cavity Flow Based on COMSOL Multiphysics

Masanori Hashiguchi[1]
[1]Keisoku Engineering System Co., Ltd.
1-9-5 Uchikanda, Chiyoda-ku, Tokyo 101-0047, Japan, hashiguchi＠kesco.co.jp

Abstract:
In the present paper, possibility of implicit large eddy simulation(LES) by using the laminar flow interface of COMSOL Multiphysics Ver.4.3a, has been examined for two-dimensional lid-driven cavity flow with the Reynolds number up to 1,000,000. The flow computations with time-dependent study step were executed stable for all cases treated here, even though no turbulence model is used. Although we need further study, the present study strongly suggests a possibility of implicit LES by using the laminar flow interface of COMSOL Multiphysics Ver.4.3a.

Keywords: Implicit LES, High Reynolds number, lid-driven cavity.

1. Introduction

Turbulence is one of the most challenging computational physics problems. Turbulence is modeled computationally by a two-stage bootstrap process [1]. The first stage, direct numerical simulation, attempts to resolve the relevant physical time and space scales but its application is limited to diffusive flows with a relatively small Reynolds number (Re). Large eddy simulation incorporates a form of turbulence modeling applicable when the large-scale flows of interest are time dependent, thus introducing common statistical models into the formulation. An alternative approach to large eddy simulation involves the use of computational fluid dynamics algorithms such as upwind scheme which introduces intrinsic subgrid turbulence models implicitly into the computed flow. This scheme is called implicit LES, and is very useful in practical flow computation because it does not need any explicit turbulence model. Unlike explicit turbulence model, implicit LES has a definite merit, that is, it needs relatively small amount of mesh system than DNS and can reproduce transition automatically from laminar flow to turbulent flow as well as DNS.

The problem of flow inside a square cavity whose lid has constant velocity has been employed to evaluate numerical methods and validate codes for solving the Navier-Stokes equations. In the works cited in Table 1 of Marchi et al.[2], the problem was solved for 11 x 11 up to 2048 x 2048node grids, and for Reynolds numbers from zero to 21,000. According to Barragy and Carey[3], the Reynolds number of 21,000 gives an upper limit where steady solution for this problem exists. Erturk and Corke[4] examined the flow up to Reynolds number of 21,000 with stream function-vorticity formulation.
Furthermore, Zhen-Hua et al.[5] also obtained solutions for Re=1,000,000 based on multi-relaxation-time Lattice Boltzmann method.

The purpose of this research is to examine whether or not the laminar flow physics interface of COMSOL Multiphysics can simulate high-Reynolds number flow based on the streamline diffusion and crosswind stabilization but not on any turbulence model. In the present paper, numerical solutions for lid-driven two-dimensional square cavity flow problem at Re=1000, 10,000, 20,000, 100,000 and 1,000,000 are obtained by time-dependent finite-element analysis of the Navier-Stokes equations without any explicit turbulence model. Through the present paper, possibility as an implicit LES will be strongly suggested when time-dependent simulation is performed.

2. Method of approach

Flow treated here is assumed to be incompressible two-dimensional and be governed by the Navier-Stokes equations and the continuity equation:

$$\rho\frac{\partial u}{\partial t} + \rho(u\cdot\nabla)u = \nabla\cdot[-pI + \mu(\nabla u + (\nabla u)^T)]\ ,$$

$$\rho\nabla\cdot \mathrm{u} = 0,$$

where ρ is the density of fluid, u=(u(x,y,t),v(x,y,t)) the flow velocity vector, p(x,y,t) the pressure, x=(x,y) the Cartesian coordinates, t the time and ∇ the nabla operator.

Figure 1 shows the lid-driven cavity with the side length of L. Top boundary is continuously moved from the left to the right with the velocity U. The other boundaries are considered as the no-slip wall. In this computation, flow is incompressible so that the pressure difference is only significant and the reference pressure level is set to zero at the bottom right corner as shown in fig.1.

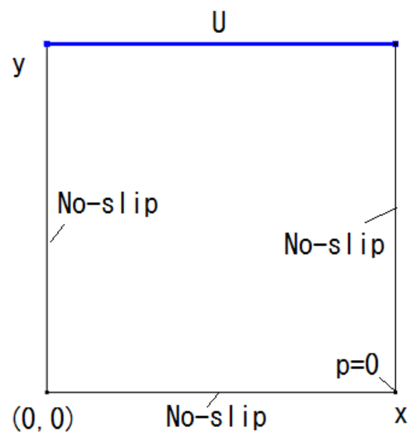


Figure 1. Schematic view of driven cavity flow.

The Reynolds number is defined as

$$\mathrm{Re} = \frac{\rho U L}{\mu},$$

where μ is the molecular viscosity of the fluid. The Reynolds number is the ratio of the nonlinear convection term and viscous term, so as the Reynolds number is increased, numerical computation becomes unstable due to an insufficient viscous diffusion and a growing nonlinear effect. Therefore we usually need a stabilization technique such as upwind scheme to damp out high-wave number nonlinear disturbance which is generated by the nonlinear convection term. In order to solve these equations, the laminar flow interface of COMSOL Multiphysics Ver.4.3a is utilized.

The stream function ψ is simultaneously obtained by solving the following PDE which is coupled to the above mentioned equations via the coefficient PDE interface[7] of COMSOL Multiphysics and with the Dirichlet condition of ψ=0 on the wall boundaries:

$$\nabla^2 \psi = -\omega,$$

where $\omega = \frac{\partial v}{\partial x} - \frac{\partial u}{\partial y}$ is the vorticity. This stream function can be used for visualization.

In this paper, numerical stabilization technique is of very importance. We utilized streamline diffusion stabilization for which COMSOL uses Galerkin least squares(GLS) method. Moreover we used cross wind diffusion which tries to smear out the boundary layer so that it becomes just wide enough to be resolved on the mesh. To obtain a sharper solution and remove the oscillation of solution, the mesh needs to be refined locally at the boundary layers. The details of these techniques are written in the manual of COMSOL Multiphysics[6]. We have to use a denser mesh near the walls where the boundary layer develops as to satisfy the no-slip condition. If we don't violate the inherent molecular viscosity effect when using numerical stabilization technique, it can be expected to realize implicit LES of flow field, in principle.

3. Results and discussion

Mesh system used here is a mapped mesh or structured type mesh. 64x64 meshes are utilized for the flow computation of Re=1,000, 128x128 for Re=10,000 and 20,000, 100,000 and 256x256 for Re=1,000,000. Figure 2 shows the mesh system used for the flow computation of Re=1,000. For the all of cases treated here, mesh is arranged as to be clustered near the wall in order to resolve boundary layer flow as shown in fig.2, and time-dependent study was executed. Dimensionless time t* defined by tU/L was up to 300 for Re=1,000, and t* up to 999.8 for Re=10,000, 20,000, 100,000 and 1,000,000.

Figure 3 shows the contours of the stream function for Re=1,000 at t*=300. This agrees with the result of the very accurate stationary computation with 601x601 uniform meshes obtained by Erturk and Corke[4] using finite-difference scheme for the stream function-vorticity formulation, although in the present

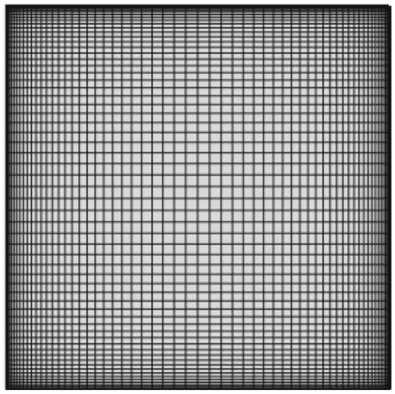


Fig.2 Mesh system of 64x64.

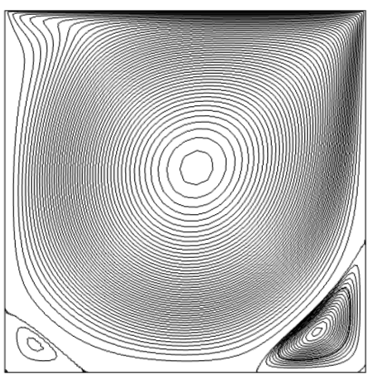
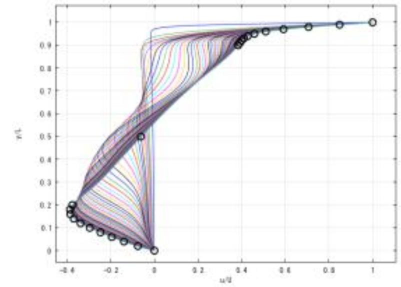


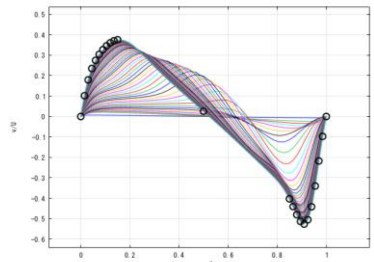
Fig. 3 Stream lines of Re=1,000 at t*=300.

computation coarse mesh is used. Figures 4 (a) and 4(b) show the time histories of x-component of velocity u along the vertical mid line, and y-component of velocity v along the horizontal mid line, respectively. As shown in fig.4, the velocity distribution approaches steady state and coincides well with the results(denoted by circles in Fig.4(a)(b)) of the stationary computation of Erturk and Corke[4] .

Figure 5 shows the contours of stream function of Re=10,000 at t*=999.8. In this case, we used 128x128 meshes and can compare the result of the very accurate computation of [4] with 601x601 uniform meshes. Secondary eddies at the upper left, lower left and lower right position are also clearly captured by the present computation as well as [4]. Figures 6(a) and 6(b)



(a) x-component.



(b) y-component.

Fig.4 Time histories of velocity distribution of Re=1,000 during t*=0.8-300

show the time evolution of x-component of velocity u along the vertical mid line, and y-component of velocity v along the horizontal

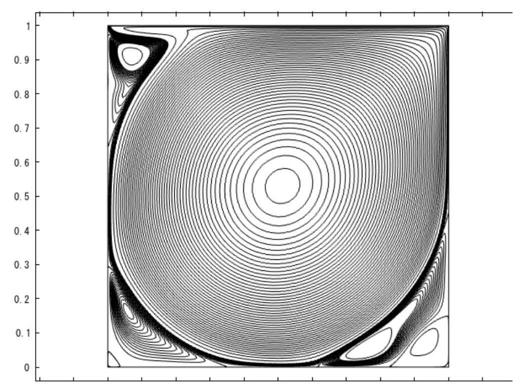
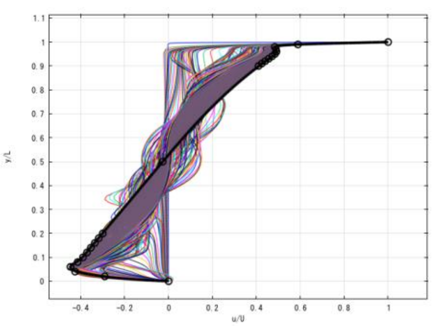


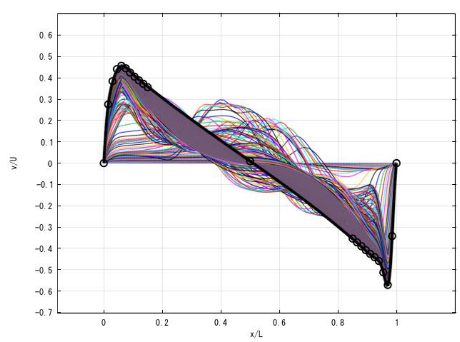
Fig.5 Stream lines of Re=10,000 at t*=999.8.

mid line, respectively. By comparing with the result of Re=1,000, it shows that the flow unsteadiness is stronger than that of Re=1,000. Up to t*=999.8, each distribution of u and v reaches steady state and coincides the stationary distributions(denoted by circles) obtained by [4].

Erturk and Corke[4] obtained by using their finite-difference scheme the solution of Re=20,000, so here we computed the case of Re=20,000. Figure 7 shows the present solution of the stream function of Re=20,000 at t*=999.8 and coincides with [4]. As well as [4], the present computation also captures the eddy (denoted by TL2 in Fig.7) clearly. Figures 8(a) and 8(b) show the resultant u and v until



(a) x-component.



(b) y-component.

Fig.6 Time histories of velocity distribution of Re=10,000 during t*=0.8-999.8.

t*=999.8 reaches steady state and coincides with the ones obtained by [4]. For clarity, the results only at t*=999.8 are shown and they coincides the steady state solution(denoted by circles) obtained by [4].

Lastly, we examined whether or not the very high Reynolds number flow of Re=100,000 and 1,000,000 can be computed by using the laminar flow interface of COMSOL Multiphysics Ver.4.3a without turbulence model as same as for low Reynolds number cases executed here. We confirmed that the time-dependent computation in the both cases runs stable up to t*=999.8 without turbulence model as same as the lower Reynolds number cases shown here.

Figures 9 (a), 9(b) show the results of the present study: vorticity contours for Re=100,000 at t*=500.8 and 999.8. Figure 10 shows the result of velocity distribution during from t*=800.8 to 999.8. During this time period, the primary vortex seems to be still and stable.

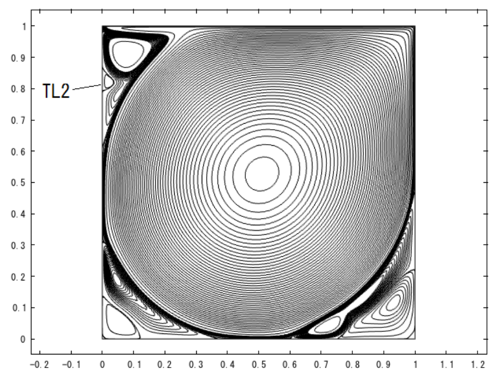
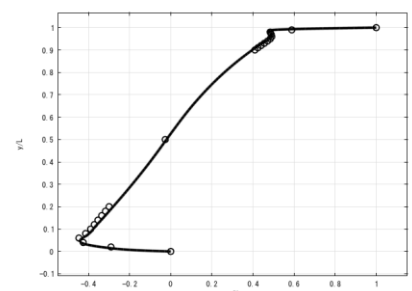
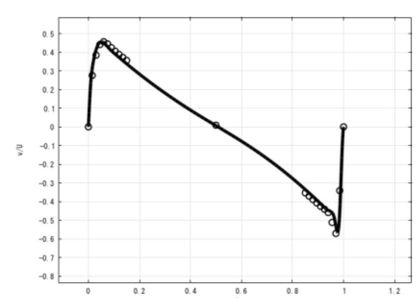


Fig.7 Stream lines of Re=20,000 at t*=999.8.



(a) x-component.

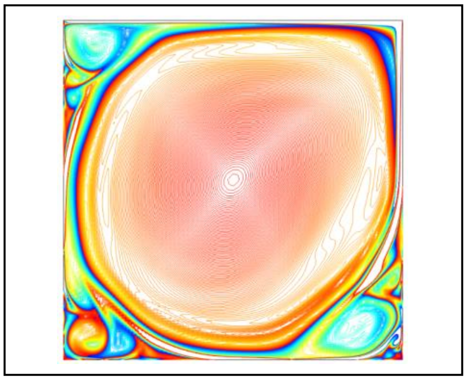


(b) y-component.

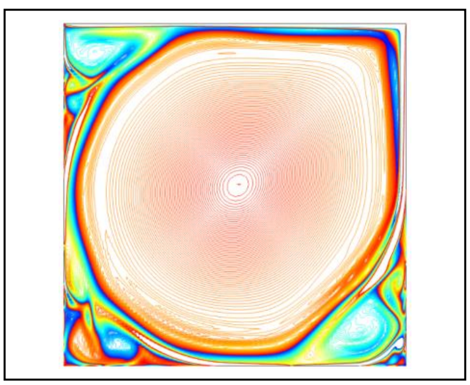
Fig.8 Velocity distribution of Re=20,000 at t*=999.8.

Figures 11(a)-(h) show the present results; the time evolution of vorticity field of Re=1,000,000. Figures 12 and 13 show the vorticity field computed for long time period.

It is a pity that there is no sufficient number of detailed literature on simulating high Re flow. According to the numerical solution obtained by the Lattice Boltzmann method of Chai Zhen-Hua et al.[5], when Re is higher than 50,000, the flow in the cavity becomes turbulent completely, even



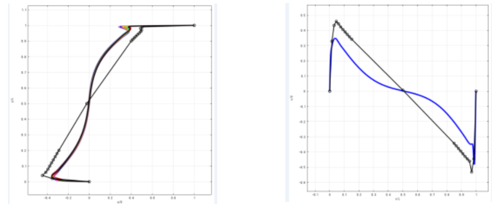
(a) t*=500.8.



(b) t*=999.8

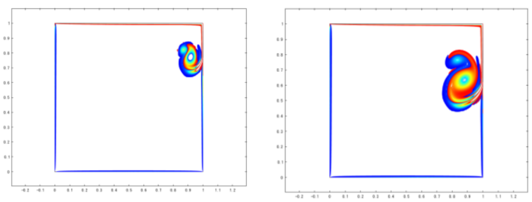
Fig.9 Instantaneous vorticity of Re=100,000.

though the plot of streamline shows a relatively primary vortex, however, the shape and the structure of flow change greatly with time. For Re as high as 500,000, there is no apparent or strict primary vortex in the middle domain. When Re is further increased to 1,000,000, many small vortices appear in the middle domain of the cavity. Based on the observation for their results near the wall, small vortices are produced from the fixed wall, which can be considered as flow separation.

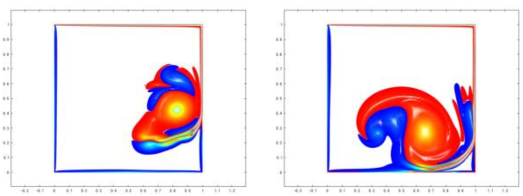


(a) x-component. (b) y-component.

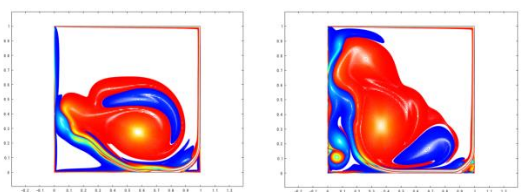
Fig.10 Velocity distributions of Re=100,000 during t*=800.8-999.8;solid line with circles for Re=20,000[4] are superimposed for comparison.



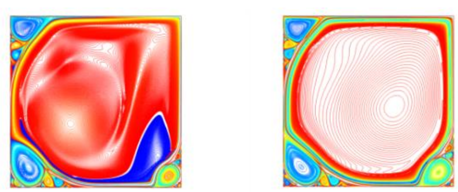
(a) t*=5.8 (b) t*=10.8



(c) t*=20.8 (d) t*=30.8



(e) t*=40.8 (f) t*=50.8



(g) t*=120.8 (h) t*=200.8

Fig.11 Time evolution of vorticity field of Re=1,000,000.

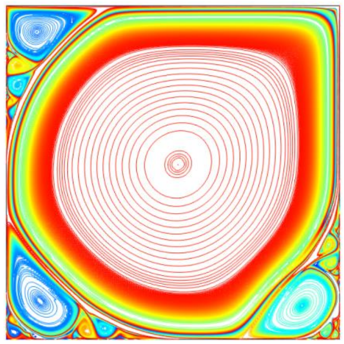


Fig.12 Vorticity field of Re=1,000,000 at t*=800.8.

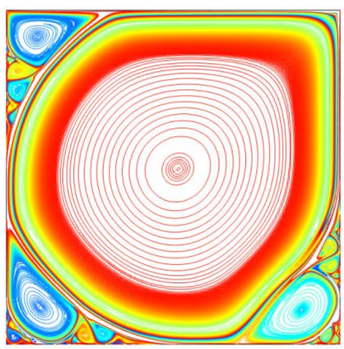
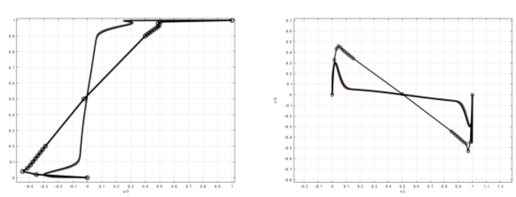


Fig.13 Vorticity field of Re=1,000,000 at t*=999.8.



(a) x-component. (b) y-component.

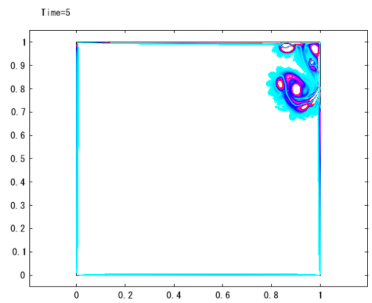
Fig. 14 Velocity distributions of Re=100,000 during t*=800.8-999.8;solid line with circles for Re=20,000[4] are superimposed for comparison.

The present results clearly shows that the series of small eddies are generated near the fixed wall surfaces like turbulent boudary layer flow, but the present primary vortex, however, seems to be stable after the long time and no penetration of small eddies into the primary vortex as reported in [5] cannot be observed. Figure 14 shows the velocity components of Re=1,000,000 during t*=800.8-999.8. Flow fluctuation is limited around the walls and the part of the primary vortex is not so fluctuated. The patterns of velocity distribution shown in Fig.14 resemble the ones obtained by [5], that is, the velocity distribution becomes differ from the rigid rotating flow as in the low Reynolds number (Re<=100,000) flow. The CPU time took about 7 hours for 256x256.

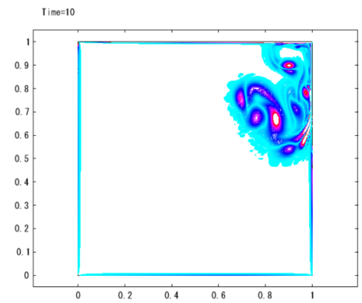
In order to examine the influence of mesh size and numerical dissipation to solution, we have done a computation of Re=1,000,000 with 400x400 meshes and less numerical diffusion which was achieved to change the tuning parameter Ck in the consistent stabilization section of the laminar flow interface to 0.1 (the default setting was 1). It was so time consuming that we have completed the unsteady computation only up to t*=100. The result shown in Fig.15, however, elucidates the fine structure of time evolution of vorticity field rather than that shown in Fig.11. Figure 16 shows streamlines at t*=100, and Fig.17 shows the resulting mid-line velocity distributions compared with those of Re=20,000[4]. These results resembles to ones visualized by the previous literature [5]. For this computation, we took 3days and 4 hours. Further computation for long time duration is proceeding now and will be published in future.

4. Conclusions

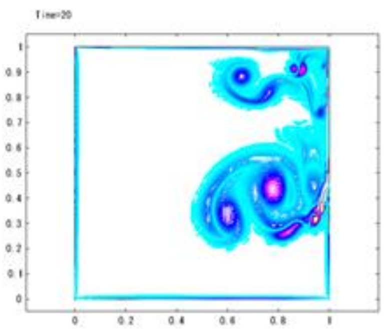
In the present paper, we investigated the flow computation of two-dimensional lid-driven cavity flow at Reynolds number up to 1,000,000 by using the laminar flow interface of COMSOL Multiphyisics 4.3a with time-dependent solver. For the Reynolds number flow cases of 1,000, 10,000 and 20,000, the present solution reached steady state and coincided with the existing literature which has used the high accurate computation with 601x601 mesh. For high Reynolds number cases of 100,000 with 128x128 mesh and 1,000,000 with 256x256 mesh, the present time-dependent computation has been executed stable without any explicit turbulence model and has reproduced small eddies around the wall and no rigid rotational



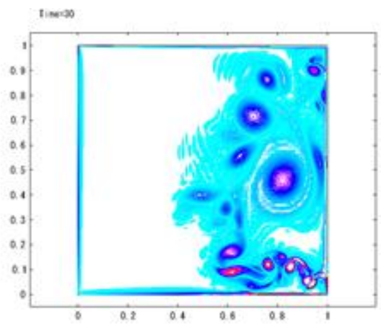
(a) t*=5



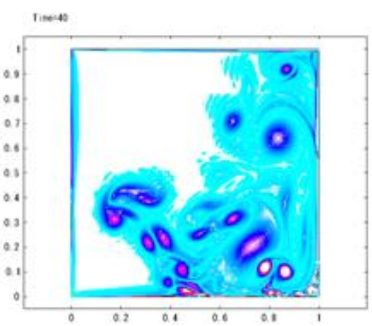
(b) t*=10



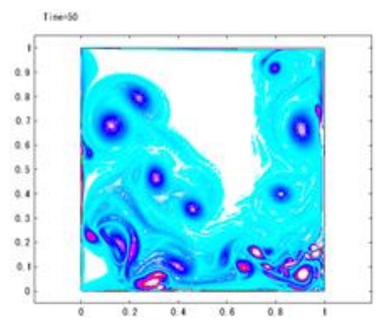
(c) t*=20



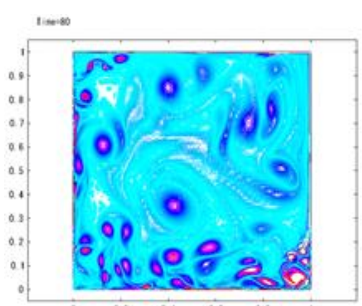
(d) t*=30



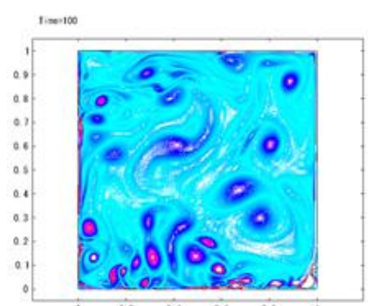
(e) t*=40



(f) t*=50



(g) t*=80



(h) t*=100

Fig.15 Time evolution of vorticity field of Re=1,000,000 at t*=5, 10, 20, 30, 40, 50, 80 and 100; this computation was done by using 400x400 meshes and less numerical diffusion.

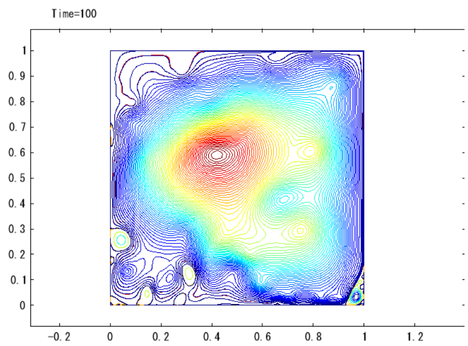


Fig.16 Instantaneous streamlines of Re=1,000,000 at t*=100 with 400x400 and less numerical diffusion.

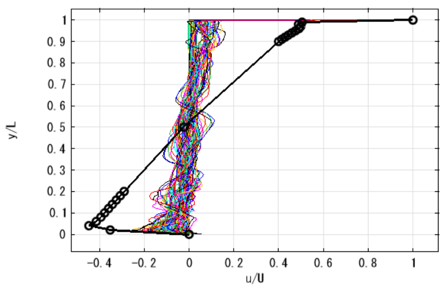


Fig. 17 Time evolution of x-component velocity of Re=1,000,000 during t*=1-100 with 400x400 and less numerical diffusion.

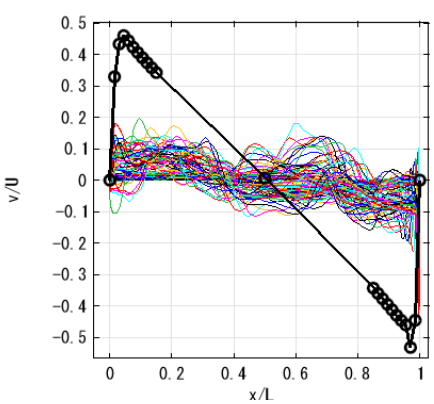


Fig. 18 Time evolution of y-component velocity of Re=1,000,000 during t*=1-100 with 400x400 and less numerical diffusion.

flow as reported previously, but no eddy penetration into the primary vortex has been observed when using coarse mesh of 256x256.

On the other hand, when using the huge mesh size of 400x400 with less numerical diffusion, we obtained many small eddies. Although we need further study, these results performed here positively suggest a possibility of implicit LES by using the laminar flow interface of COMSOL Multiphysics without any explicit turbulence model. On the other hand, more effort on the memory reduction and speed-up based on, e.g., regional decomposition is strongly desired.

References

[1] J.P. Boris et al., New Insights Into Large Eddy Simulation, Fluid Dyn. Res. 10(1992) 199-288.

[2] C.H.Marchi et al.: The lid-driven square cavity flow: numerical solution with a 1024x1024 grid, J.Braz.Soc.Mech.Sci.& Eng.,vol.31,no.3,2009.

[3] E.Barragy and G.F.Carey: Stream function-vorticity driven cavity solution using p finite elements, Computers & Fluids, Vol.26, no.5,1997, pp.453-468.

[4] E.Erturk et al., Numerical Solutions of 2-D Steady Incompressible Driven Cavity Flow at High Reynolds Numbers, Int.J.Numer.Meth.Fluids 2005; 48:747-774.

[5] Chai Zhen-Hua et al., Simulating high Reynolds number flow in two-dimensional lid-driven cavity by multi-relaxation –time lattice Boltzmann method, Chinese Physics, Vol.15, No.8, 2006,1855-1863.

[6] COMSOL Multiphysics 4.3a, COMSOL Multiphysics, Reference guide.

[7] Bjorn Shodin: New Design Paradigm of Multiphyisics Simulation Platform, Proc. of Comp. Eng. Conf., vol.17, 2012.