SIMPLE SOLVER FOR DRIVEN CAVITY FLOW PROBLEM

Vaidehi Ambatipudi
Department of Mechanical Engineering
Purdue University
West Lafayette, Indiana 47906
Email: vaidehi@purdue.edu

ABSTRACT

This report presents the solution to the Navier-Stokes equations. Standard fundamental methods like SIMPLE and primary variable formulation has Been used. The results are analyzed for standard CFD test case- Driven cavity flow. Different Reynold numbers and grid sizes have been studied. The results match very well with results from a benchmark paper.

NOMENCLATURE

V Velocity Vector.
u u velocity.
v v velocity.
p scalar pressure.
S_u Source in u momentum equation.
S_v Source in v momentum equation.
i x direction unit vector.
j y direction unit vector.
f body force.
V volume.
F Flow rate.
b Source term in discrete equation.
Re Reynolds number.
ρ density.
μ diffusion constant.
∇ Divergence Operator.
ϕ unit quantity in general transport equation.
Γ Diffusion coefficient.
J flux.
A Area vector.
u_b boundary velocity.
u corrected velocity.
a_{nb} neighbor coefficient.
u_{nb} neighbor velocity.
α_p pressure under-relaxation.
α momentum equation under-relaxation.
u^* guess velocity.

INTRODUCTION

Many numerical methods for solving the 2D Navier- Stokes equation in the literature are tested using the 2D driven cavity problem. In this course project SIMPLE algorithm is used with primitive variables velocity and pressure. The reference paper uses the multigrid method and vorticity stream function formulation. The use of simple iterative techniques to solve th Navier-Stokes equations might lead to slow convergence. The rate of convergence is also generally strongly dependent on parameters such as Reynolds number and mesh size.

PROBLEM DEFINITION

The standard benchmark in literature for testing 2D Navier-Stokes equations is the driven cavity flow problem. The problem considers incompressible flow in a square domain (cavity) with a upper lid moving with a velocity u as shown in Fig.1.The other boundaries have no-slip tangential and zero normal velocity boundary condition. The main goal is to obtain the velocity field in steady state from the NS equations. Vorticity stream function formulations can be used which results in only two equations but it is difficult to derive boundary conditions. Primitive variable formulation is preffered these days.

GOVERNING EQUATIONS AND BOUNDARY CONDITIONS

The governing equations are those of 2D incompressible Navier - Stokes equations, continuity and u and v momentum

 Copyright © by ASME

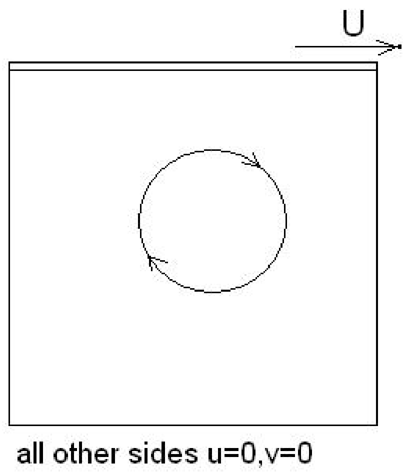

Figure 1. Driven Cavity Flow in a square domain

equations.

$$\nabla.(\rho\mathbf{V}) = 0 \quad (1)$$

$$\nabla.(\rho\mathbf{V}u) = \nabla.(\mu\nabla u) - \nabla p.\mathbf{i} + S_u \quad (2)$$

$$\nabla.(\rho\mathbf{V}v) = \nabla.(\mu\nabla v) - \nabla p.\mathbf{j} + S_v \quad (3)$$

The source term for an Newtonian fluid can be simplified into

$$S_u = f_u + \frac{\partial}{\partial x}\left(\mu\frac{\partial u}{\partial x}\right) + \frac{\partial}{\partial y}\left(\mu\frac{\partial v}{\partial x}\right) - \frac{2}{3}\frac{\partial}{\partial x}(\mu\nabla.\mathbf{V}) \quad (4)$$

$$S_v = f_v + \frac{\partial}{\partial y}\left(\mu\frac{\partial v}{\partial y}\right) + \frac{\partial}{\partial x}\left(\mu\frac{\partial u}{\partial y}\right) - \frac{2}{3}\frac{\partial}{\partial y}(\mu\nabla.\mathbf{V}) \quad (5)$$

Comparing equation 2 to the General Scalar transport equation

$$\frac{\partial(\rho\phi)}{\partial t} + \nabla.\rho\mathbf{V}\phi = \nabla.\rho\mathbf{V}\phi = \nabla.(\Gamma\nabla\phi) + S \quad (6)$$

we can see that

$$\phi = u \quad (7)$$

$$\Gamma = \mu \quad (8)$$

$$S = S_u - \frac{\partial p}{\partial x} \quad (9)$$

and hence the discretization techniques discussed in class can be used for calculating gradients and other terms. The difficulty in solving these equations is that the NS equations are nonlinear and the pressure in the domain is unknown. The continuity and momentum equations are also coupled partial differential equations and need to be solved sequentially.

NUMERICAL METHOD

Discretization

The equation 2 can also be written as

$$\nabla\mathbf{J} = S \quad (10)$$

which on integrating over the control volume and using divergence theorem to get

$$\sum\mathbf{J}.\mathbf{A} = S.\Delta V \quad (11)$$

where

$$J = \rho\mathbf{V}u - \mu\nabla\phi \quad (12)$$

$$S = -\frac{\partial P}{\partial x} \quad (13)$$

Co-located storage of the pressure and velocity variables at the cell centres leads to the problem of checkerboarding. This is because the cell centre values of pressure and velocity get cancelled out on expanding the face gradient terms. To overcome this problem staggered grid has been used for discretization of the momentum equations. The staggered grid for the u momentum equation is shown in Fig.2 along with the neighboring velocity vectors for calculation of velocity gradients. Staggerred grid in vertical direction is used for v momentum equation. Pressure is stored on the original grid and the pressure difference terms are evaluated as a difference of cell centre pressure values.

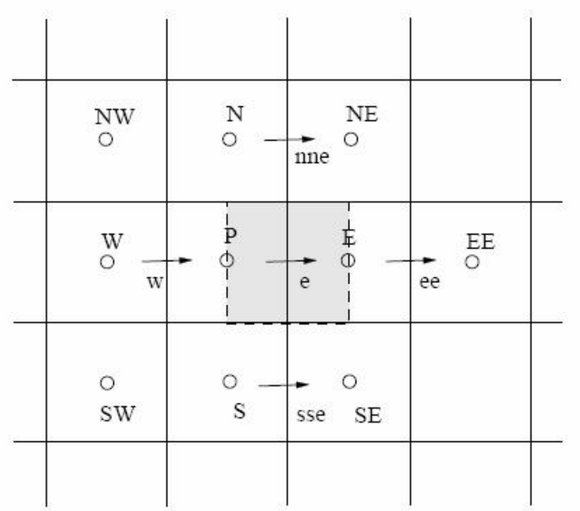

Figure 2. Neighbors for Ue momentum control volume

Copyright © by ASME

The evaluation of the flux term on the east face is shown below.

$$A = \Delta y.\mathbf{i} \tag{14}$$

$$\mathbf{J}.\mathbf{A}_{eastface} = \rho {u_E}^* u_E \Delta y - \mu \Delta y \left(\frac{u_{ee} - u_e}{\Delta x} \right) \tag{15}$$

$$u_E = \left(\frac{u_e + u_{ee}}{2} \right) \tag{16}$$

The face value of velocity on the face of the staggered grid is evaluated as a Central Difference(CDS) of the neighbors. This assumption is reasonable since in a driven cavity flow due to appearance of vortices there is no upwind direction of the velocities. We can evaluate the face flux on the west face similarly. The equations below show the calculation of face flux on the north face. The only difference is that the convection term consists of a u-v term. The v_N term is evaluated by central differencing again.

$$A = \Delta x.\mathbf{j} \tag{17}$$

$$\mathbf{J}.\mathbf{A}_{northface} = \rho {v_N}^* u_N \Delta x - \mu \Delta x \left(\frac{u_{nne} - u_e}{\Delta y} \right) \tag{18}$$

$$v_N = \left(\frac{v_N + v_{NE}}{2} \right) \tag{19}$$

The face flux on the south face can be evaluated accordingly. The source term from 13 is evaluated as the difference in cell centre pressure multiplied by the volume of the cell.

$$-\left(\frac{P_E - P_P}{\Delta x} \right) \Delta x \Delta y \tag{20}$$

Discretization of boundary cells

Although, the velocity boundary condition is used in calculating gradients in the first cell using the staggered grid for the u-momentum discretization it doesnot consider momentum balance on the boundary strip Fig.3. The last(far east) staggered cell is also only half Δx thick. Similarly momentum balance is not considered on the south boundary strip. Seperate momentum balance equations can be written for the boundary strips but including the boundary strip into the first cell is convenient. In the momentum equations for boundaries there is no change in the diffusion terms.

$$\frac{\partial u}{\partial x} = \frac{u_{ee} - u_e}{\Delta x} \tag{21}$$

$$\frac{\partial u}{\partial x} = \frac{u_e - u_b}{\Delta x} \tag{22}$$

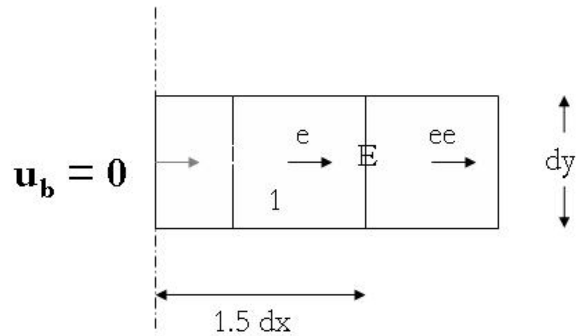

Figure 3. Neighbors for Ue momentum control volume

The convection term on the west face will be zero due to the zero velocity boundary condition whereas the convection term on the east side is calculated normally $\rho \Delta y {u_E}^* u_E$ Also since the boundary strip is included the cross section area in the vertical direction and total volume of this cell change. Instead of including the boundary strip into the first cell we could also write seperate momentum conservation equations for the boundary slip.

$$A = 1.5 \Delta x \hat{j} \tag{23}$$

$$V = 1.5 \Delta x \Delta y \tag{24}$$

SIMPLE Solver Algorithm

Semi-Implicit Method for Pressure-Linked Equations was first proposed by Patankar and Spalding(1972). Here we start with the discrete continuity equation and substitute into this the discrete u and v momentum equations containing the pressure terms resulting in a equation for discrete pressures.SIMPLE actually solves for a relative quantity called pressure correction. We guess an initial flow field and pressure distribution in the domain. The set of momentum and continuity equations are coupled and are nonlinear so we solve the equations iteratively. The pressure field is assumed to be known from the previous iteration. Using this the u and v momentum equations are solved for the velocities. At this stage the newly obtained velocities dont satisfy continuity since the pressure field assumed is only a guess. Corrections to velocities and pressure are proposed to satisfy the discrete continuity equation.

$$u = u^* + u' \tag{25}$$

$$v = v^* + v' \tag{26}$$

$$p = p^* + p' \tag{27}$$

where u^* v^* and p^* are the guess values and u' v' and p' are the corrections. The simple algorithm also requires the corrected velocities and pressure to satisfy the momentum equations leading

Copyright © by ASME

to the corrected momentum equations.

$$a_e u'_e = \sum_{nb} a_{nb} u'_{nb} + \Delta y (p'_P - p'_E) \quad (28)$$

$$a_n v'_n = \sum_{nb} a_{nb} v'_{nb} + \Delta x (p'_S - p'_P) \quad (29)$$

Approximations to the velocity correction are made by ignoring the $\sum_{nb} a_{nb} u'_{nb}$ and $\sum_{nb} a_{nb} v'_{nb}$. Substituting these corrected velocities into the continuity equations yields a discrete pressure correction equation.

$$a_P p'_P = \sum_{nb} a_{nb} p'_{nb} + b \quad (30)$$

$$a_E = \rho_e d_e \Delta y \quad (31)$$

$$a_W = \rho_w d_w \Delta y \quad (32)$$

$$a_N = \rho_n d_n \Delta x \quad (33)$$

$$a_S = \rho_s d_s \Delta x \quad (34)$$

$$a_P = \sum_{nb} a_{nb} \quad (35)$$

$$b = F^*{}_w - F^*{}_e + F^*{}_s - F^*{}_n \quad (36)$$

where $d_i = \frac{\Delta y}{a_i}$ and $F_i^* = \rho u_i{}^* \Delta y$ for e, w, n and s. Here the scarborough condition is satisfied only in equality. The figure below shows the steps involved in arriving at a converged solution.

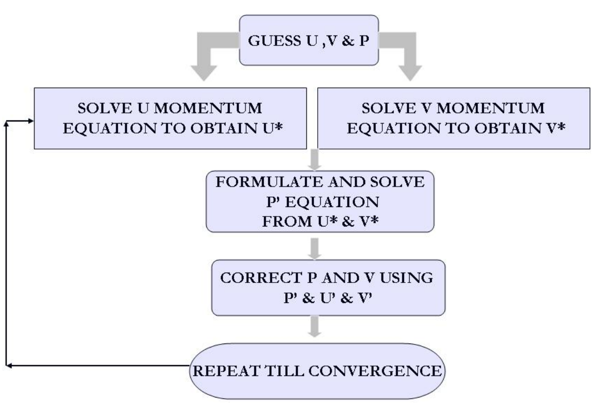

Figure 4. Flow chart showing the SIMPLE Algorithm

Under-relaxation

The velocity corrections are approximated by dropping the velocity part of the corrected momentum equations which places the entire burden of the velocity correction on pressure correction. Large pressure corrections might lead to poor pressure iterates so the pressure correction is under-relaxed to correct p^*.It is necessary to under-relax the momentum equations due to the nonlinear nature of the equations.

$$p = p^* + \alpha_p p' \quad (37)$$

$$\frac{a_e u_e}{\alpha} = \sum a_{nb} u_{nb} + \Delta y (P_P - P_E) + \frac{(1-\alpha)}{\alpha} a_e u_e{}^* \quad (38)$$

In this problem the under-relaxation values for the pressure and momentum respectively are 0.1-0.3 and 0.7 depending on the Reynolds number of the problem.

RESULTS

A uniform grid is assumed in the x and y direction. The momentum equations are discretized and the SIMPLE algorithm is implemented. Various grid sizes have been studied and for different Reynold numbers. The velocity solutions are compared to the results quoted in the paper. The graphs include computed u-velocity along the vertical centre line and v- velocity along the horizontal centre line compared to the discrete information given in the reference. Here all the plots show results of the finest mesh ie. 150X150. Also the stream lines have been plotted for each Re value and are compared to the stream lines shown in the reference. Figures 5,6,7 show the velocity plots for Re=400. Here we can see that computed values match very well with the reference values. We can also see that the stream line plots show

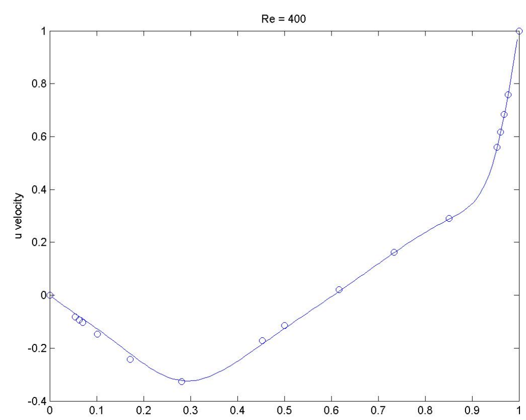

Figure 5. $Re = 400 U velocity Grid 150 X 150$

good match. Apart from a primary vortex we see the formation of secondary vortices on the corners of the domain. The next three figures show results for higher Re of 1000. Here also there is good agreement with the reference values.Also for higher Re values the primary vortex shifts more towards the centre of the

Copyright © by ASME

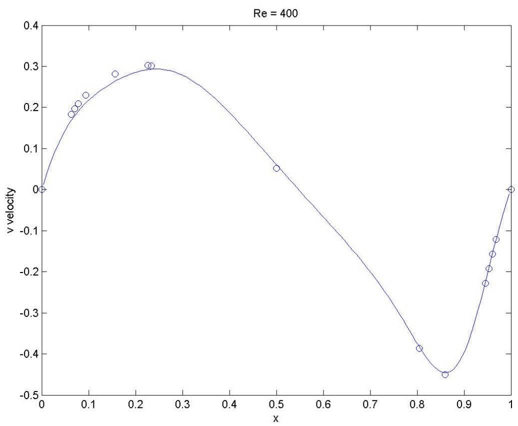

Figure 6. $Re = 400 V velocity Grid 150 X 150$

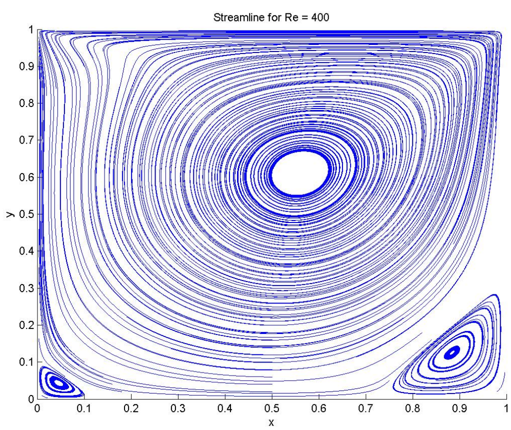

Figure 7. $Re = 400 Streamlines Grid 150 X 150$

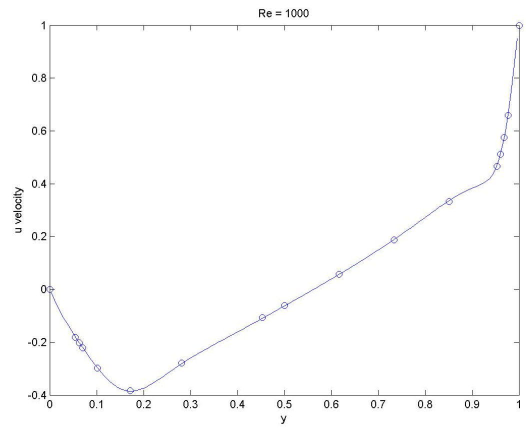

Figure 8. $Re = 1000 U velocity Grid 150 X 150$

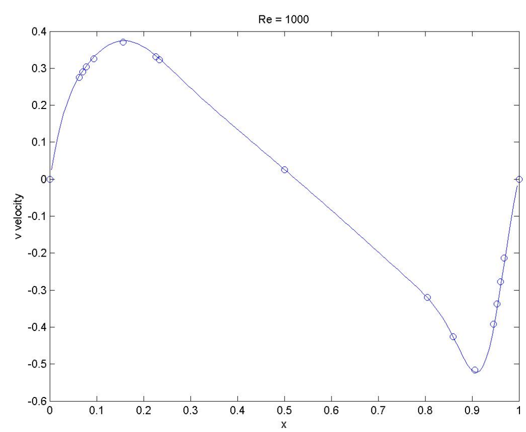

Figure 9. $Re = 1000 V velocity Grid 150 X 150$

domain. The results for other Re also can be seen in subsequent pages. For higher Re values the primary vortex shifts more to the centre and more corner secondary vortices are formed. The secondary vortices are also convected towards the centre of the domian for higher Re values. Also with the convection of secondary vortices more vortices are formed at the corners.
The stream line function plots for various Re can be verified with the plots shown in the section ADDITIONAL FIGURES.

CONCLUSIONS

Comparing the results with those of the benchpark paper on driven cavity flow show that SIMPLE solver is adequate to solve complex flow field problems like the driven cavity flow. There is a good match with of the computed results with the reference values. Fine details like the corner vortices are also accurately predicted using fine grids. Other than some minor computational difficulties the SIMPLE solver is very efficient in solving flow problems. The accuracy and convergence might be increased using refined techniques like SIMPLER and SIMPLE-C though.

REFERENCES

U.Ghia, K N Ghia and C.T . Shin 1982. High Re solutions for Incompressible flow using the Navier-Stokes equations and a Multigrid method. Journal of computational physics 48, 387-411.

Other references

ME608 class notes
Maciej Matyka, Solution to 2 D incompressible NS equations with SIMPLE,SIMPLER and Vorticity stream function approaches.Driven Lid Cavity problem:Solution and Visualization.

Copyright © by ASME

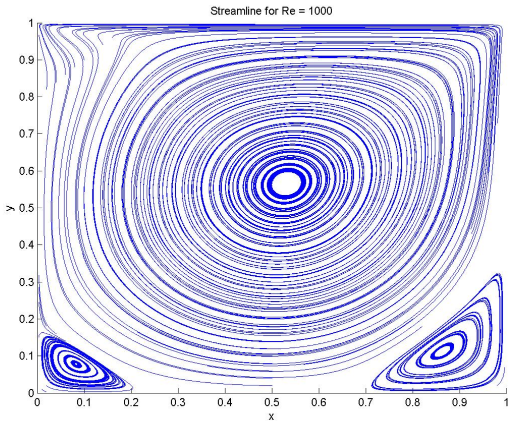

Figure 10. $Re = 1000StreamlinesGrid150X150$

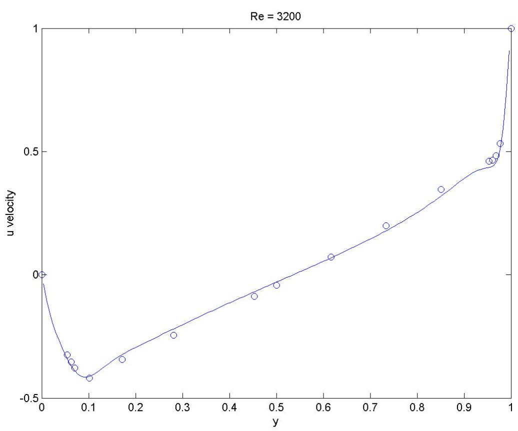

Figure 11. $Re = 3200UvelocityGrid150X150$

APPENDIX A:ADDITIONAL FIGURES

These additional figures show the stream line contours in the reference paper. We can see a very close resemblance with the computed stream line solutions.

APPENDIX B: Code

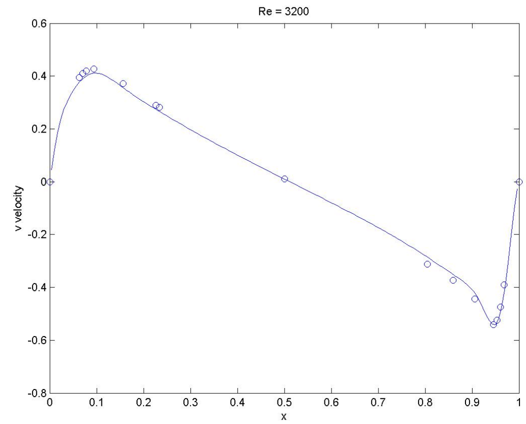

Figure 12. $Re = 3200VvelocityGrid150X150$

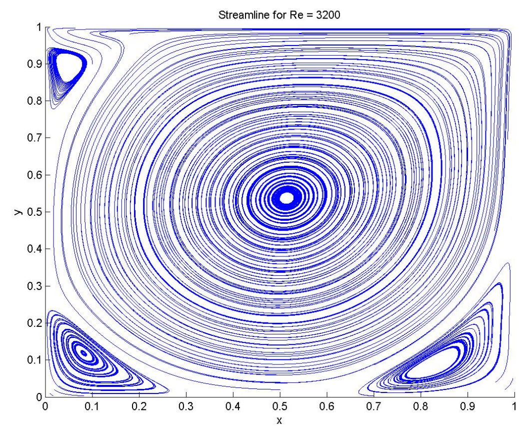

Figure 13. $Re = 3200StreamlinesGrid150X150$

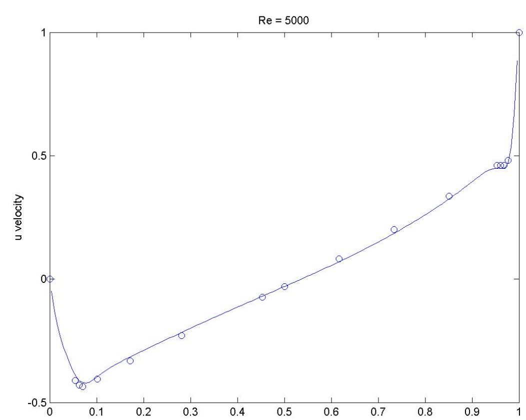

Figure 14. $Re = 5000UvelocityGrid150X150$

Copyright © by ASME

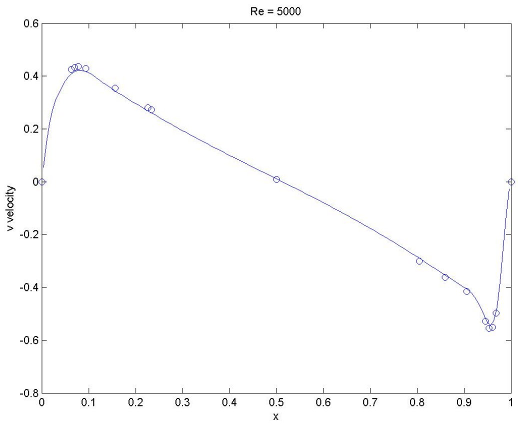

Figure 15. $Re = 5000 V velocity Grid 150 X 150$

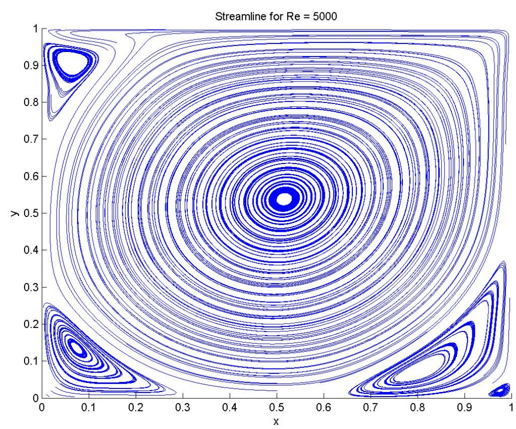

Figure 16. $Re = 5000 Streamlines Grid 150 X 150$

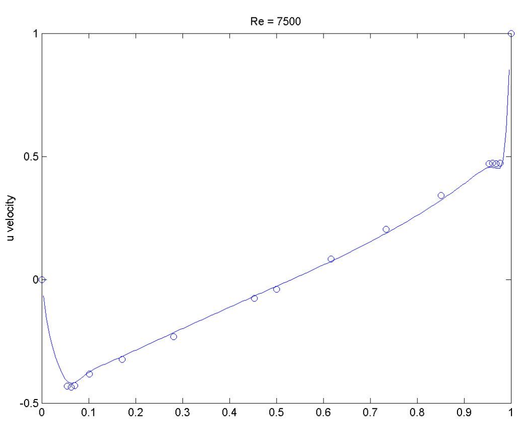

Figure 17. $Re = 7500 U velocity Grid 150 X 150$

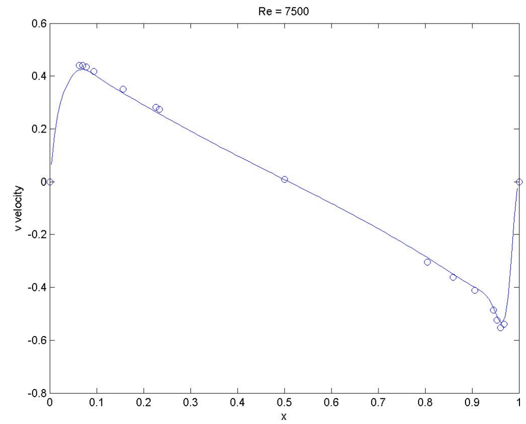

Figure 18. $Re = 7500 V velocity Grid 150 X 150$

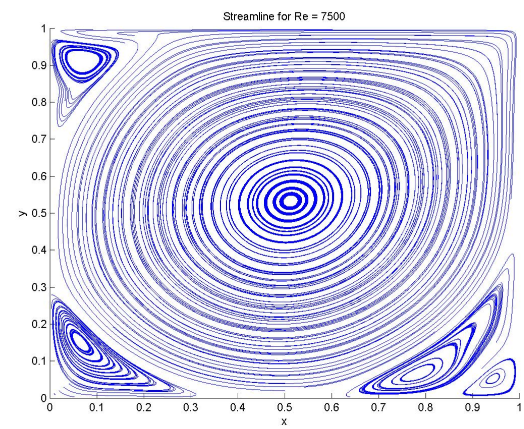

Figure 19. $Re = 7500 Streamlines Grid 150 X 150$

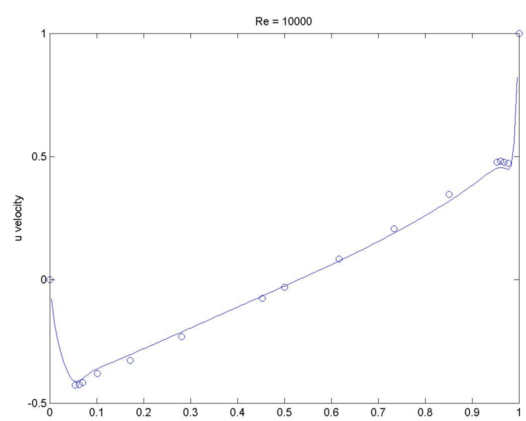

Figure 20. $Re = 10000 U velocity Grid 150 X 150$

Copyright © by ASME

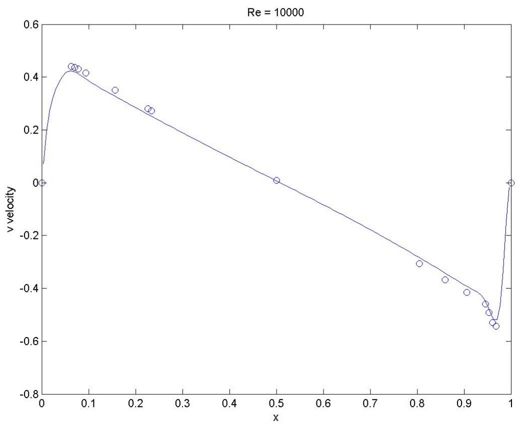

Figure 21. $Re = 10000 V velocity Grid 150 X 150$

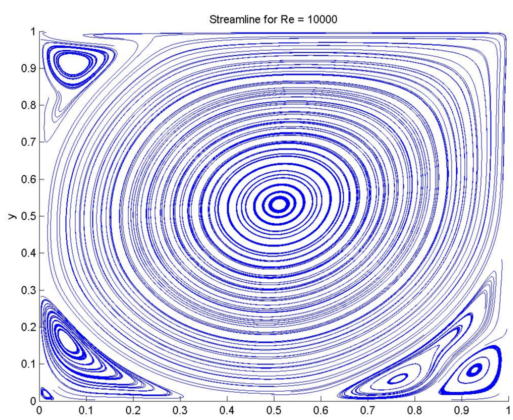

Figure 22. $Re = 10000 Streamlines Grid 150 X 150$

Copyright © by ASME